

NASA Contractor Report 194465

1N-39
2937
115 P

Turbomachinery Forced Response Prediction System (FREPS): User's Manual

M.R. Morel
Sverdrup Technology, Inc.
Lewis Research Center Group
Brook Park, Ohio

and

D.V. Murthy
University of Toledo
Toledo, Ohio

Prepared for
Lewis Research Center
Under Contract NAS3-25266
and Grant NAG3-1669



National Aeronautics and
Space Administration

(NASA-CR-194465) TURBOMACHINERY
FORCED RESPONSE PREDICTION SYSTEM
(FREPS): USER'S MANUAL Final Report
(Sverdrup Technology) 115 p

N94-29104

Unclass

63/39 0002937

100

100

100

100

100

100

100

100

100

100

100

100

100

100

100

100

Preface

This report is the User's Manual for the turbomachinery **FORced REsponse Prediction System** (FREPS), version 1.2. The code is capable of predicting the aeroelastic behavior of axial-flow turbomachinery blades. This document is meant to serve as a guide in the use of the FREPS code with specific emphasis on its use at NASA Lewis Research Center (LeRC). A detailed explanation of the aeroelastic analysis and its development is beyond the scope of this document, and may be found in the references. FREPS has been developed by the NASA LeRC Structural Dynamics Branch.

Please note that the FREPS code is being made available strictly as a research tool. Neither NASA LeRC, nor any contractors, nor grantees that have contributed to the code development, assume liability for application of the code beyond research needs.

The manual is divided into three major parts: an introduction, the preparation of input, and the procedure to execute FREPS. Part I includes a brief background on the necessity of FREPS, a description of the FREPS system, the steps needed to be taken before FREPS is executed, an example input file with instructions, presentation of the geometric conventions used, and the input/output files employed and produced by FREPS. Part II contains a detailed description of the command names needed to create the primary input file that is required to execute the FREPS code. Also, Part II has an example data file to aid the user in creating their own input files. Part III explains the procedures required to execute the FREPS code on the Cray Y-MP, a computer system available at the NASA LeRC.

Any questions or related items concerning this computer code can be directed to Dr. D. V. Murthy at the NASA Lewis Research Center, Cleveland, OH 44135 or call (216) 433-6714.

Acknowledgement

This work was supported by the Structural Dynamics and the Launch Vehicle Technology Branches at NASA Lewis Research Center (grant NAG3-1449 and contract NAS3-25266). George Stefko was the technical monitor and James Gauntner was the project manager. Also, a special thanks to Daniel Hoyniak for his recommendations and contributions to the FREPS project.

TABLE OF CONTENTS

Preface	i
Acknowledgement	i
TABLE OF CONTENTS	ii

Part I - INTRODUCTION

1.1	Scope	1.1-1
1.2	FREPS - An Overview	1.2-1
	1.2.1 Objectives and Assumptions	1.2-1
	1.2.2 Execution Overview	1.2-1
1.3	FREPS Input Example	1.3-1
1.4	FREPS Theory & Flowchart	1.4-1
1.5	FREPS Geometric Conventions and Units	1.5-1
1.6	FREPS Input/Output File Structure	1.6-1
	1.6.1 FREPS Input Files	1.6-1
	1.6.2 FREPS Output Files	1.6-2

Part II - FREPS INPUT PREPARATION

2.1	General Description	2.1-1
2.2	FREPS Keywords	2.2-1
2.3	Sample Input Data File	2.3-1

TABLE OF CONTENTS

Part III - PROCEDURE TO RUN FREPS

3.1	Introduction	3.1-1
3.2	Run FREPS on the Cray Y-MP	3.2-1

Part IV - REFERENCES

Part V - APPENDICES

Appendix A: Nomenclature	A-1
Appendix B: MSC/NASTRAN	B-1
Appendix C: SFLOW	C-1
Appendix D: LINFLO	D-1
Appendix E: Outline of FREPS Output	E-1

INTRODUCTION

1.1 Scope

Future advances in propulsion system technology are directed toward increased fuel economy, extended life, and reduced operating costs. In the design of aerospace propulsion systems, the demand to produce high power in light-weight turbomachines while operating under stringent safety and reliability requirements is a challenging task. This is especially true with the advent of new materials, complex and novel blade geometries, and the extreme loading and environmental conditions. The reliability of engine designs is generally insured by expensive and time-consuming engine testing. As a supplement or alternative to engine testing, blade designers should employ advanced numerical tools to predict the dynamic blade response at operating conditions. If such computational methods are to successfully supplement or supplant engine testing, then their predictions must account for the complex interaction between multi-disciplinary effects (*e.g.* fluid/structure) to ensure reliable turbomachinery designs. In addition, the numerical tools must be efficient enough to be used in a design environment. In this spirit, a computer program called FREPS (Forced REsponse Prediction System) was developed under the Earth-to-Orbit Propulsion Technology Program to predict the vibratory response of turbomachinery blades.

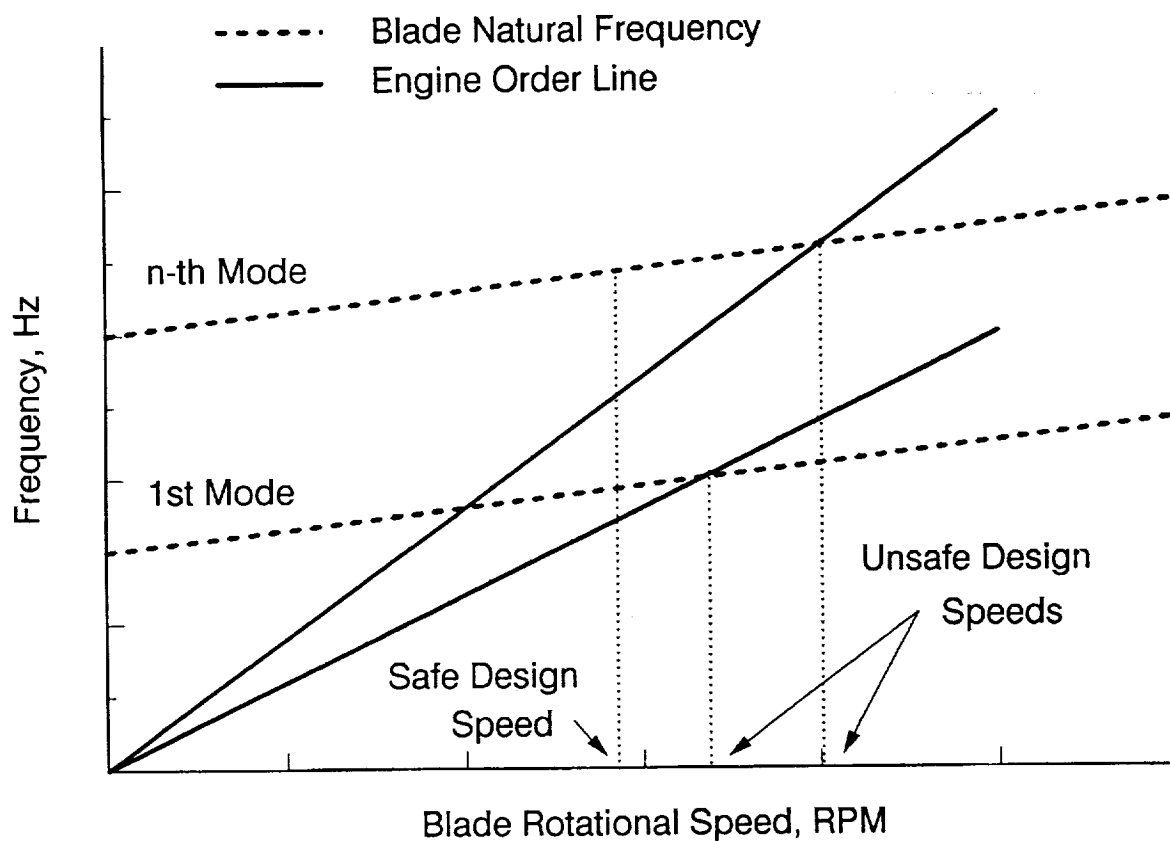
Turbomachinery blades operate in an inherently dynamic environment and are subject to numerous unsteady excitations (*e.g.*, aerodynamic and mechanical excitations). The forced dynamic response of turbomachinery blading to these unsteady excitations is an important cause of fatigue in many aircraft engines. The inability to predict fatigue failures directly relates to increased engine development costs, operational restrictions, and elevated maintenance costs. Controlling the forced vibratory response of turbomachinery blades is crucial for reliable operation of rocket and aircraft engines. Over the past 30 years, serious engine failures have been attributed to fatigue induced by forced response.

The excitations causing forced vibratory response in turbomachinery blades cannot be removed by redesign because they arise from the very nature of the operation of turbomachinery. Blade rows in turbomachinery rotate relative to each other and are intrinsically subject to time-varying disturbance fields. Generally, the turbomachinery designer is not free to reduce the excitation levels and the only recourse is to design for acceptable vibratory response stresses for given excitation levels. Design choices at this stage are difficult because of the complexity of the unsteady flow field and the intricacies of interaction between the aerodynamics and structural dynamics. Designing for acceptable dynamic stresses usually results in increased design cost, reduced engine performance or increased weight. In order to improve this critical design process, accurate and timely predictions of the quantitative forced response of the blades, in

terms of stress amplitudes, frequencies, principal stress directions, and critical locations, is of vital importance. Thus, an advanced tool, such as FREPS, for the prediction of the dynamic response will be extremely useful in the design of turbomachinery blading.

The prevailing design procedure for avoiding forced dynamic response problems is partly engineering and partly art. Traditionally, the Campbell diagram has been used to provide a safe margin between the blade resonant frequency and the integral orders of engine speed having potentially significant excitation. The use of the Campbell diagram is schematically illustrated in Figure 1.1-1. However, this approach is only able to indicate potential forced response problems and provides no quantitative information about the vibratory stress levels. The interpretation of the Campbell diagram and key design decisions have generally been very empirical. For example, damping plays no role in the Campbell diagram approach so that there is no guarantee that this technique is conservative. This can be particularly dangerous in the case of the newer engines with innovative designs and advanced material systems, where prior experience and insights do not apply.

Although the Campbell diagram technique has proven very successful over the past few decades, the push for higher stage loading and lighter engine weight raises many more forced response problems. In addition, the blades can undergo significant fatigue damage even when the excitation frequency is not very close to one of the natural frequencies due to low damping, high frequency, and high excitation levels. Furthermore, no quantitative information is available for life prediction and lack of this knowledge is a significant barrier in these cases. The drawbacks of the Campbell diagram approach are reflected in long development cycles, expensive testing procedures, and turbomachines that do not provide the desired life. Therefore, the goal of FREPS is to create a system which will enable a designer to quantitatively predict the turbomachinery response due to the many inherent sources of excitation.

Figure 1.1-1: Typical Campbell Diagram for an Engine Turbine Blade

1.2 FREPS - An Overview

The fundamental goal in developing the FREPS system was to provide a state of the art quantitative forced response prediction tool. This would enable turbomachinery designers to improve the fatigue characteristics of the blades in a systematic manner. The computer program is an efficient forced response prediction tool for all types of axial-flow turbomachinery. Blade structural dynamic, steady and unsteady aerodynamic analyses are efficiently integrated to calculate the forced vibration response amplitudes in turbomachinery blades to a variety of excitations.

1.2.1 Objectives and Assumptions

The specific objectives of the FREPS system are:

- 1.) Calculate the aerodynamic damping (hence, determine the aeroelastic stability) of a turbomachinery blade row at given incident flow conditions.
- 2.) Predict the maximum principal stresses in turbomachinery blading for given mechanical excitation forces and incident gust excitations.

Aeroelastic stability is determined by considering the equilibrium of elastic, inertial and damping forces of structural as well as aerodynamic origin. An eigenvalue problem is formulated and the resultant damping is obtained and checked. If the resultant damping is negative, then the blade row is aeroelastically unstable. If the resultant damping is positive, then the forced response problem may be solved to calculate the maximum principal stresses due to mechanical or aerodynamic excitations.

The amplitudes and interblade phase angles associated with mechanical excitation forces and incident gusts must be specified by the user. The actual excitation forces on the blade due to wake excitations are calculated within the code.

All blades are assumed to be identical. So are all blade passages. The structural model is a three-dimensional finite-element model. The steady and unsteady aerodynamic models are two-dimensional. The steady mean flow is calculated from the nonlinear velocity potential equation. The unsteady flow is assumed to be a small perturbation to the steady mean flow resulting in a linearized unsteady aerodynamic model that accounts for the effects of thickness, camber, incidence and distortion of incoming gust through blade passage.

1.2.2 Execution Overview

The FREPS system integrates a structural dynamic analysis with steady and unsteady aerodynamic analyses to perform an aeroelastic analysis. The analyses are all implemented in separate modules shown in Figure 1.2-1. Arrows from block to block display data flow and code dependencies. The different analysis modules communicate with one another through database files. The steady aerodynamic and structural dynamic analyses are executed separately (refer to Table 1.2-1), taking input data in their own required format, using meshes appropriate to the particular analysis and producing output databases. Once the steady aerodynamic analysis is complete, the mesh required for the unsteady aerodynamic analysis (called by FREPS internally) is calculated and stored in a database. The various databases are used by the FREPS executive modules to provide input for performing the integrated aeroelastic analysis.

The finite element model and the input for the steady flow analysis are created from the blade geometry and material properties. The steady flow results are used as input to generate a streamline mesh required for unsteady flow analysis. After these three analyses are performed properly, the FREPS code can be executed to predict the forced dynamic stresses due to aerodynamic and mechanical excitations. A flutter analysis of the blade can also be performed at the same time or run as a separate case.

In summary, the main contributing technologies to FREPS are the steady and unsteady aerodynamic analyses and the structural dynamic analysis. As a result, three stand-alone computer codes must be executed before the FREPS code can be executed. These are:

- 1.) **MSC/NASTRAN [1]** - commercial finite element code to compute the natural frequencies and mode shapes
- 2.) **SFLOW [2]** - full potential steady flow solver to solve for the steady flow field
- 3.) **LINFLO [3-4]** - linearized inviscid unsteady analysis to generate the mesh for the unsteady aerodynamic analysis of the cascade (LINFLO is also called from inside FREPS).

The output file of each of these runs are used as input to the FREPS code. Again, displayed in Table 1.2-1 are the necessary steps to be performed by the user before executing the FREPS code.

First, a finite element model of the component must be developed to perform the structural dynamic analysis. For example, shown in Figure 1.2-2 is a typical blade structure and finite element model. FREPS is capable of reading the output from MSC/NASTRAN which is used to perform the structural dynamic analysis. Both the model and results from the MSC/NASTRAN structural analysis are needed as input for FREPS. The dynamic modal stresses are also required if a forced response analysis is requested by the user. The user is referred to Appendix B for further information on the MSC/NASTRAN model and analysis.

The three-dimensional (3-D) structural analysis from MSC/NASTRAN and two-dimensional (2-D) aerodynamic analysis (SFLOW and LINFLO) are integrated using a strip theory procedure. Thus, the present aeroelastic analysis is a quasi-3-D approach. The 2-D airfoil shapes (Figure 1.2-2) that are defined in the SFLOW input for the steady aerodynamic analysis are associated in the FREPS input file with "strips" defined by specifying nodes from the finite element model. This association is used to combine the 3-D structural model with the 2-D aerodynamic models to perform a quasi-3-D aeroelastic analysis.

The steady flow field is calculated prior to the unsteady flow analysis in order to account for the coupling of the steady and unsteady flow. Therefore, SFLOW is executed to determine the steady compressible flow along each strip using velocity potential equations. The SFLOW database files created from the analysis are used in the linearized unsteady potential flow theory implemented in LINFLO. This linearized potential approach produces another set of database files which, along with the database files created by SFLOW, are needed as input to the FREPS code. As a result, the user must first generate SFLOW and LINFLO input decks in order to execute the respected codes. Located in Appendices C and D are more information on creating the input for these routines.

Along with the output from MSC/NASTRAN, SFLOW, and LINFLO, a primary FREPS input file is also required. A quick reference of the data needed to prepare this file is located in Table 1.2-2. The primary input file defines a specified number of two-dimensional strips from the finite element model (see Figure 1.2-2). Each strip is assumed to be a stream surface of constant radius and constant thickness. For the completion of the aeroelastic model, blade row properties, such as the number of blades and hub and tip radii, are defined. The fluid, thermodynamic, and aerodynamic properties are specified for each strip. Also, if forced response is required the amplitude and phase of the mechanical excitation forces or the incoming gust are defined.

Table 1.2-1: Preliminary Steps to be Performed Before FREPS can be Executed

Stand-Alone Codes to be Executed Prior to FREPS	Required Files for FREPS
MSC/NASTRAN (Finite Element Code) - Define Finite Element Model - Perform Structural Dynamic Analysis	Standard Output File - Nodes and Coordinates - Element Connectivity - Eigenvalues and Eigenvectors Punch File of Modal Stresses - Dynamic Stresses in Binary Form
SFLOW (Steady Potential Flow Solver) - Define Airfoil - Cascade Description - Define Flow Mesh	Database File - One File for Each Strip
LINFLO (Unsteady Flow Analysis) - Input Database Files from SFLOW - Mesh Parameters	Database File - One File for Each Strip

Table 1.2-2: Information Required to Perform Stability and/or Forced Response Analyses in FREPS

Stability Analysis	Forced Response Analysis
Airfoil Strips Strips defined by listing the nodal points from the finite element model	
Fluid and Thermodynamic Properties - Temperature - Pressure - Density - Velocity - Specific heat ratio	
Aerodynamic Properties - Inlet fluid velocity - Inlet flow angle - Inlet Mach number - Strip aeroelastic stream surface height - Stagger or rotation angle	
Database and Finite Element Result Files	
Structural Damping Ratio	
Rotor - Number of blades on rotor - Speed of rotor - Tip radius - Hub radius	
Interblade Phase Angle All possible phase angles or define a specified number of them	
Natural Frequency Modes Modes retained in analysis	Excitation Forces - Harmonic mechanical force - Gust forcing function
Flutter Frequency Appropriate or actual flutter frequency	Modal Stress File Results stored in file from free-vibration analysis
	Elements and Nodes Define elements where principal stresses are calculated Define nodes where displacements are calculated

Figure 1.2-1: Structure of the FREPS System

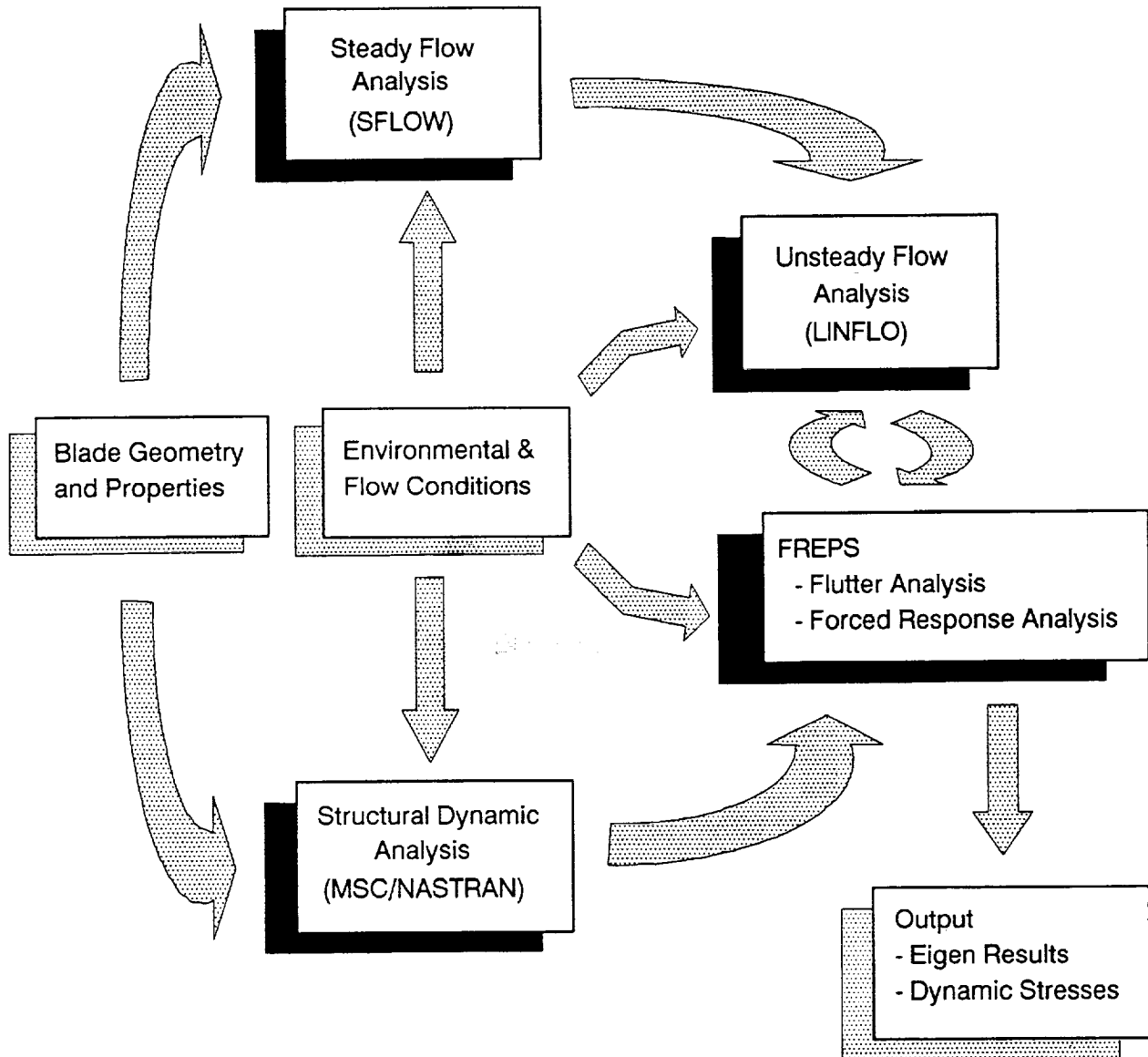
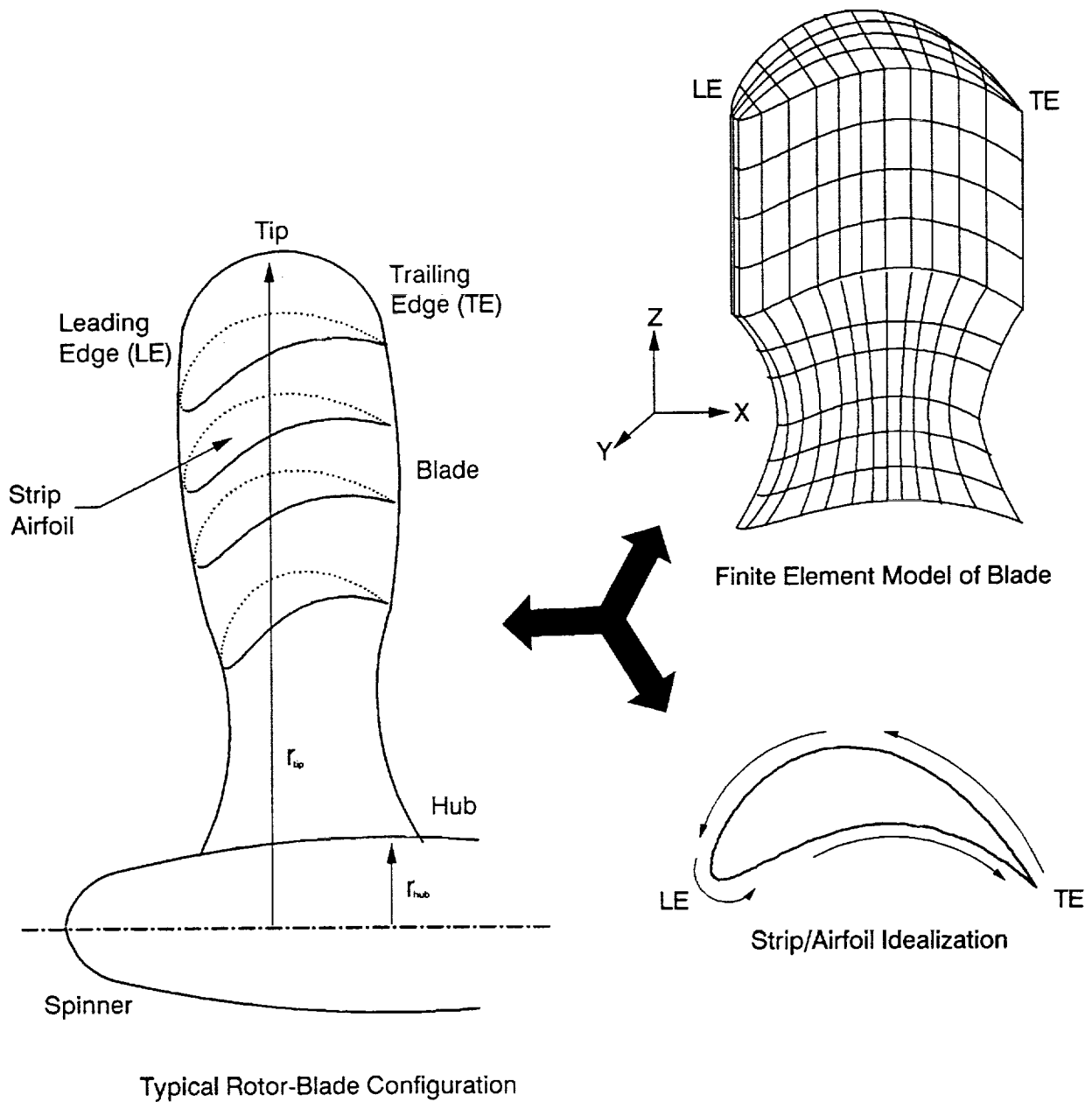


Figure 1.2-2: Typical Rotor-Blade, Finite Element Model, and Airfoil Configuration



1.3 FREPS Input Example

A sample case with a step-by-step description of the procedure, shown in Table 1.3-1, to prepare the FREPS primary input file (Table 1.3-2) is provided in this section. For this example, it is assumed that the preliminary steps outlined in Table 1.2-1 in the previous section have been completed. Briefly, perform the free-vibration analysis (MSC/NASTRAN) of the finite element model depicted in Fig. 1.3-1 and predict the unsteady aerodynamic behavior (SFLOW-LINFLO) of the fluid contained in the blade flow passages. The following is a list of the output file names from executing MSC/NASTRAN, SFLOW, LINFLO prior to executing FREPS:

- 1.) mscnastran.output
- 2.) airfl_1.U18, airfl_2.U18, airfl_3.U18, airfl_4.U18
- 3.) airfl_1.U17, airfl_2.U17, airfl_3.U17, airfl_4.U17

The objective of this example case is to determine the blade row aeroelastic stability at design speed of a turbine blade.

After all preliminary steps have been completed, a primary FREPS input file is generated, refer to Table 1.2-2 (pg. 1.2-4) for a review of the input data required for the analysis. The finite element model of the turbine blade, shown in Figure 1.3-1, is divided into four radial strips of finite height along the span of the blade from the hub to the tip. The locations of the strips should be chosen to "best" represent the modal displacements from the free-vibration analysis. For example, this blade experiences significant modal displacement in the tip region of the blade in the first mode, as a result, a strip was chosen in the upper 10% of the blade. The following criteria (the Z direction is assumed to correspond with the span length from hub to tip) was used in choosing the four strips:

Strip Number	Radial Zone Location (in.)
1	1.25 - tip
2	1.00 - 1.25
3	0.75 - 1.00
4	0.50 - 0.75

One strip was picked from each of the four zones and are defined in the FREPS input deck by listing the nodal points along the exterior of the model. The corresponding nodal geometry is extracted from the MSC/NASTRAN results to form the shape of the airfoil. FREPS assumes that the airfoil shape coordinates are plotted in the X-Y plane, refer to Figure 1.2-2. Also from the MSC/NASTRAN results, the first four natural frequencies and mode shapes are

used as input.

The working fluid is air, operating at standard atmospheric conditions, and was assumed to obey the perfect gas law, values used for this example are located in Table 1.3-3. In order to associate the finite element airfoil (strip) coordinates and LINFLO airfoil coordinates a rotation angle must be defined. This angle is equivalent to the stagger angle that is used for the steady and unsteady flow results. Also, the strip aerodynamic properties are listed in Table 1.3-4.

For the description of the rotor, the turbine blade described above is one of 50 blades on the rotor. The rotor is moving at a speed of 6000.0 revolutions per minute (rpm). This model assumes that the rotor vibrates as a "tuned" rotor such that every blade in the cascade has identical frequencies and mode shapes.

The individual steps, along with the corresponding line numbers in the example input data file, (Table 1.3-2), that are required to model this problem are presented in Table 1.3-1. Line numbers are provided in the sample input deck for reference purposes only and are not part of the input file. In the input deck, a maximum of 80 spaces per line are available and blank spaces and/or commas are used to separate the fields on a line.

Table 1.3-1: Procedure in Preparing an Input Data File

Step No.	Line No.	Description of Data
1	1	The first record in the input file is the TITLE record, which gives a brief description of the problem being solved.
2	2-7 and throughout	The \$ is the comment record. These commented lines explain further the problem being solved. Also, provide a brief description for the command records throughout the input deck.
3	16-23 26-33 36-44 & 46-53	The STRIP record is used to create the stream-surface strips. In the second field on the line is the strip identification (stripID) number with the corresponding exterior nodes listed from the finite element model, refer to Fig. 1.3-1. The numbering sequence of the nodes should start at the upper trailing edge and move counter-clock-wise around the model, i.e. suction- side to pressure-side. For the users convenience in reading the nodes, the list of nodes are distributed on eight STRIP records for each individual strip. A total for four separate strips are used. Each strip has 53 nodes outlining its airfoil shape. The strips are numbered from tip to hub with the first strip closest to the tip and the fourth strip located near the hub. The strip nodes must begin and end with the same node.
4	58	The fluid in which the blade operates in, is defined by the SFLUID record. For this example, all blades operate in the same working fluid, therefore, only one SFLUID record is necessary. The second field in the SFLUID record is the fluid strip identification (fluidID) number (100) followed by the working fluid properties, refer to Table 1.3-3 for these values and the order in which they appear. Only two of T, p, or ρ are required because the fluid is assumed to behave as a perfect gas.
5	61-64	In order to link the finite element coordinates and the LINFLO airfoil coordinates, a rotation angle is required. The SROTATE record defines the strip ID number and the angle of rotation between these two coordinate systems. The finite element model is on the X-Y coordinate system and the LINFLO coordinate system is on the ξ - η , as a result, the rotation angle is equivalent to the stagger angle (θ) and these values are located in Table 1.3-4.

Step No.	Line No.	Description of Data
6	67-70	The SAERO record defines the aerodynamic strip properties. Starting with the second field, the aeroelastic strip identification number (aeroID) is given followed by the inlet fluid velocity, inlet flow angle, inlet Mach number, and the aeroelastic stream surface height. These values are located in Table 1.3-4. Each strip has a particular set of aerodynamic properties, therefore, four SAERO record exists each with a unique aeroID.
7	73-76	The SGROUP record defines the relationships between the group strips (groupID), strips (stripID), and property (e.g. aeroID and fluidID) identifications. The second field identifies the groupID number, the second field the stripID number followed by the aeroID number and the fluidID number.
8	81-84	To link the LINFLO and SFLOW database files to the proper strip, the DATABASE record is employed. After the command record, the second field contains the stripID number with the file name of the database file immediately following. An assumption is made when using this command; separate database files exist for LINFLO and SFLOW having the same file name but different file types, e.g., SFLOW files are stored under *.U18 and LINFLO files are filed under *.U17. In this example, each strip has a unique database file assigned to it.
9	89	The FERESULT record signifies the type of FE results file (NASTRAN) and the file name (mscnastran.output) of the output.
10	100	The ROTOR record defines the rotor (a blade row, in general) that the turbine blade operates on. The second through fifth fields accommodate the number of blades (50) on the rotor, the speed of the rotor in rpm (6000.0), the hub radius (0.5 in), and the tip radius (1.5 in).
11	111	The AESET record activates the SGROUP records defined previously. For this example, all SGROUPs are active for the aeroelastic analysis, as a result, groupIDs 100, 200, 300, 400 are displayed.
12	113	In order to define which normal modes to use in the analysis, the AEMODES record defines the normal modes retained in the predictions. The first four natural modes are used for this example.

Step No.	Line No.	Description of Data
13	118	The AMSAVE record saves the calculated aerodynamic matrices to a user specified file. A UNIX file name (AM.aero) is defined after the keyword.
14	130	The AEFREQ record defines the assumed flutter frequency. The assumed flutter frequency can be assigned in two different ways. First, a natural frequency can be assumed, as is in this case (1, which identifies the first free-vibration frequency), or secondly, the user can estimate a reasonable value.
15	132-134	AESIGMA defines the interblade phase angles for the flutter analysis. The allowable angles are: the angle (7.2°) obtained by dividing 360° by the number of blades (50) defined in the ROTOR record and its integer multiples. This value describes the phase angle difference between the motion of adjacent blades on the rotor. For this example, every other phase angle was chosen, 0.0° , 14.4° , 28.8° ,, 345.6° .
16	136	The FLUTTER record initiates the flutter analysis procedure within FREPS.
17	146	The OUTPUT record controls the output sent to the FREPS primary output file. If no values or zeros are assigned, the default options are used.
18	147	The PLOTOUT record controls the results that can be plotted or written to an external file. The user has the option of generating an extra file containing the raw data or use the IMSL routines to generate line plots within the primary output file. In this case, the raw data is requested. Therefore, the second field contains the data type 'GENERIC' and the third field contains the file name of the raw data (raw.data).
19	154	The END record indicates the end of the primary FREPS input.

Table 1.3-2: FREPS Sample Input

```

Line 1 . . . . . 80
No.
1  TITLE    TURBINE BLADE -- FLUTTER ANALYSIS
2  $=====
3  $          TURBINE BLADE AEROELASTIC ANALYSIS
4  $          THIS JOB ANALYZES THE BLADE USING 4 RADIAL STRIPS FOR THE
5  $          AIRFOIL SECTION OF THE BLADE.
6  $
7  $=====
8  $
9  $
10 $-----> STRIP DEFINITION <-----
11 $-----
12 $
13 $ Strips are numbered in a counterclockwise direction starting at the
14 $ trailing edge and moving along the suction-side.
15 $
16 STRIP  1      3      27      75      159      243      327      411
17 STRIP  1      495     579     663     747     831     915     999
18 STRIP  1     1083    1206    1321    1394    1397    1401    1405
19 STRIP  1     1409    1413    1417    1421    1424    1427    1428
20 STRIP  1     1426    1422    1419    1414    1410    1406    1402
21 STRIP  1     1398    1318    1203    1080     996     912     828
22 STRIP  1       744     660     576     492     408     324     240
23 STRIP  1       156      72      26        3
24 $
25 $
26 STRIP  2        8      37      95     179     263     347     431
27 STRIP  2     515     599     683     767     851     935    1019
28 STRIP  2     1103    1226    1341    1448    1547    1665    1764
29 STRIP  2     1767    1853    1897    1901    1904    1907    1908
30 STRIP  2     1906    1902    1899    1894    1850    1662    1544
31 STRIP  2     1445    1338    1223    1100    1016     932     848
32 STRIP  2       764     680     596     512     428     344     260
33 STRIP  2       176      92      36        8
34 $
35 $
36 STRIP  3        13      47     115     199     283     367     451
37 STRIP  3     535     619     703     787     871     955    1039
38 STRIP  3     1123    1246    1361    1468    1567    1685    1793
39 STRIP  3     1878    1951    2003    2036    2053    2058    2052
40 STRIP  3     2030    1993    1934    1867    1784    1682    1564
41 STRIP  3     1465    1358    1243    1120    1036     952     868
42 STRIP  3       784     700     616     532     448     364     280
43 STRIP  3       196     112      46      13
44 $
45 $
46 STRIP  4        18      60     135     219     303     387     471
47 STRIP  4     555     639     723     807     891     975    1059
48 STRIP  4     1143    1266    1381    1492    1593    1711    1810
49 STRIP  4     1893    1912    1916    1919    1922    1923    1921
50 STRIP  4     1917    1914    1909    1872    1789    1690    1575
51 STRIP  4     1479    1375    1263    1140    1056     972     888
52 STRIP  4       804     720     636     552     468     384     300
53 STRIP  4       216     132      56      18

```

```

54 $
55 $ FLUID AND AERO DESCRIPTION FOR STRIPS
56 $
57 $ fluidID T(°R) p(psi) ρ(lbm/in³) a(fps) γ
58 SFLUID 100 518.69, 14.69, 0.0000, 1116.4, 1.4
59 $
60 $ stripID 0
61 SROTATE 1 0.0
62 SROTATE 2 0.0
63 SROTATE 3 0.0
64 SROTATE 4 0.0
65 $
66 $ aeroID W β M H
67 SAERO 100, 0.0, 114.0, 0.3800, 0.225
68 SAERO 200, 0.0, 63.0, 0.3850, 0.250
69 SAERO 300, 0.0, 57.0, 0.4300, 0.250
70 SAERO 400, 0.0, 55.0, 0.4500, 0.775
71 $
72 $ groupID stripID aeroID fluidID
73 SGROUP 100 1 100 100
74 SGROUP 200 2 200 100
75 SGROUP 300 3 300 100
76 SGROUP 400 4 400 100
77 $
78 $ STRIP DATABASE DEFINITIONS
79 $
80 $ stripID filename
81 DATABASE 1 airfl_1
82 DATABASE 2 airfl_2
83 DATABASE 3 airfl_3
84 DATABASE 4 airfl_4
85 $
86 $ DEFINE FE RESULTS FILE
87 $
88 $ Fetype filename
89 FERESULT NASTRAN mscnastran.output
90 $
91 $-----
92 $-----> END STRIP DEFINITION <-----
93 $-----
94 $
95 $*****
96 $*****> ROTOR DESCRIPTION <*****
97 $*****
98 $
99 $ NofBLADES Ω(rpm) hubRADIUS tipRADIUS
100 ROTOR 50 6000.0 0.5 1.5
101 $
102 $*****
103 $*****> END ROTOR DESCRIPTION <*****
104 $*****
105 $
106 $*****
107 $*****> AEROELASTIC SET DEFINITIONS <*****
108 $*****
109 $
110 $ groupIDs
111 AESET 100 200 300 400
112 $ natural modes
113 AEMODES 1 2 3 4

```

1.3 FREPS Input Example

```

114 $
115 $ save aerodynamic matrices
116 $
117 $ filename
118 AMSAVE AM.aero
119 $
120 $
121 $
122 $
123 $
124 $
125 $
126 $
127 $
128 $ assumed flutter frequency, 1st mode
129 $ mode f(hz) w(rad/sec)
130 AEFREQ 1, 0.00, 0.00
131 $ n,  $\sigma$ , interblade phase angles
132 AESIGMA 11 0.0 14.4 28.8 43.2 57.6 72.0 86.4 100.4 115.2 129.6 144.0
133 AESIGMA 11 158.4 172.8 187.2 201.6 216.0 230.4 244.8 259.2 273.6 288.0 302.4
134 AESIGMA 3 316.8 331.2 345.6
135 $ type of analysis
136 FLUTTER
137 $
138 $
139 $
140 $
141 $
142 $
143 $
144 $
145 $ IP1 IP2
146 OUTPUT 0 0
147 $ data_type filename
148 PLOTOUT GENERIC raw.data
149 $
150 $
151 $
152 $
153 $
154 $

```

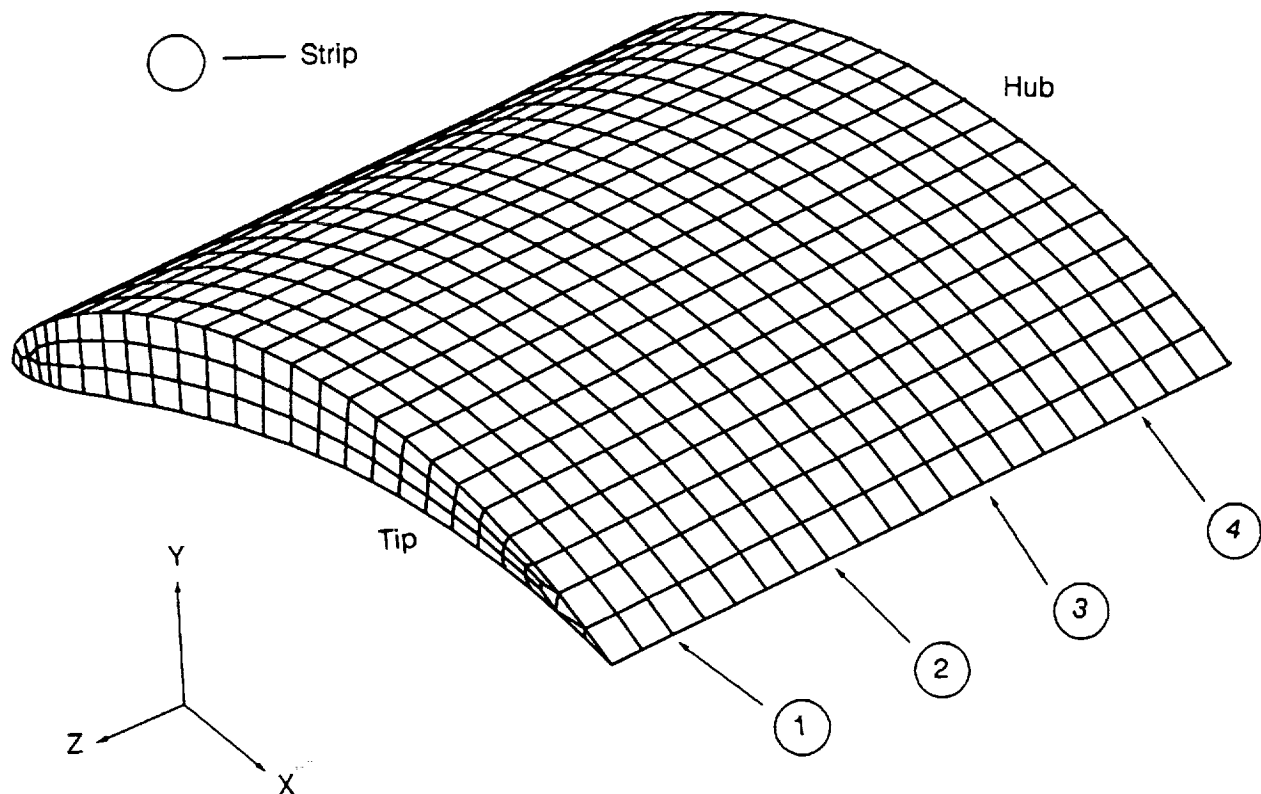

Table 1.3-3: Working Fluid Properties

Temperature, T (°R)	Pressure, p (psi)	Density, ρ (lbm/in ³)	Velocity, a (fps)	Specific Heat Ratio, γ
518.69	14.69	0.4424×10^{-4}	1116.4	1.40

Table 1.3-4: Turbine Aeroelastic Strip Properties

Strips	Stagger Angle θ (°)	Flow Angle β (°)	Mach Number M	Stream Surface Height H (in.)
1	0.0	114.0	0.3800	0.275
2	0.0	63.0	0.3850	0.250
3	0.0	57.0	0.4300	0.250
4	0.0	55.0	0.4500	0.775

Figure 1.3-1: Finite Element Model of the Blade and Aeroelastic Strip Definition



1.4 FREPS Theory & Flowchart

FREPS models the dynamic response problem in terms of dynamically linearized equations of motion specifying the equilibrium between inertial, damping, elastic, unsteady aerodynamic and mechanical excitation forces. Three-dimensional structural analysis and two-dimensional aerodynamic analysis are integrated using a strip theory procedure. Thus, the present aeroelastic analysis use a quasi-three-dimensional approach. The fundamental task in FREPS is to find the solution to the standard finite element representation of the dynamic response problem given by

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{F}_A^u(\mathbf{u}(t), \dot{\mathbf{u}}(t), t) + \mathbf{F}_A(t) + \mathbf{F}_M(t) \quad (1)$$

where $\mathbf{u}$ represents the vector of global degrees of freedom of the blade row and $\mathbf{M}$, $\mathbf{C}$, and $\mathbf{K}$ are respectively the mass, damping, and stiffness matrices of the blade row. On the right hand side of eq. (1) are the forcing functions, $\mathbf{F}(t)$. The excitation forces of aerodynamic origin are denoted by subscript A and those of mechanical origin by subscript M. The aerodynamic forces are split into motion-dependent and motion-independent forces. The superscript u denotes the motion-dependent component of the aerodynamic forces. The motion-dependent aerodynamic forces modify the structural stiffness and damping matrices. All matrices and vectors are of size $N \times m$ where N is the number of blades on the blade row and m the number of degrees of freedom per blade.

The customary assumption that the blade row is tuned, *i.e.*, all the blades and passages in the row are identical, is made to take advantage of cyclic symmetry and reduce the problem size to that of one blade and one passage. This assumption is obviously an approximation because in reality all blades on a blade row are never completely identical. However, the tuned rotor assumption allows considerable simplification in the formulation when complex structural and aerodynamic models are employed. Moreover, the influence of mistuning can be investigated by means simpler than a full analysis modeling each blade separately (*e.g.*, see reference 6).

The motion $u_j(t)$ of the j -th blade can be expressed in terms of constant inter-blade phase angle modes as

$$u_j(t) = \sum_{r=1}^N u^r(t) e^{i\beta_r j} \quad (2)$$

where $u^r(t)$ denotes motion in the mode where adjacent blades vibrate with a constant phase angle β_r between them. Cyclic symmetry implies that this phase angle can take only the N values given by

$$\beta_r = \frac{2\pi r}{N} \quad (3)$$

Cyclic symmetry also implies that the constant inter-blade phase angle modes are uncoupled allowing us to reduce the size of the problem to that of one blade sector only (as opposed to an entire blade row). At present, FREPS assumes the rotor disk to be rigid, allowing us to ignore structural coupling among blades. Using the transformation given by eq. (2), the equations of motion can now be reduced to a form identical to eq. (1), except that all matrices and vectors are now of size m and one set of such equations exists for each of the N values of β_r . Thus, for the r -th interblade phase angle mode, the equations of motion can be written as

$$\mathbf{M}\ddot{\mathbf{u}}^r + \mathbf{C}\dot{\mathbf{u}}^r + \mathbf{K}\mathbf{u}^r = \mathbf{F}_A^r(\mathbf{u}^r(t), \dot{\mathbf{u}}^r(t), t) + \mathbf{F}_A^r(t) + \mathbf{F}_M^r(t) \quad (4)$$

The matrices $\mathbf{M}$, $\mathbf{C}$, and $\mathbf{K}$ are now the mass, damping and stiffness matrices respectively of the blade rather than the blade row. The superscript r will be dropped in later equations for simplicity. Modal representation of the blade was employed in FREPS to further reduce the number of degrees-of-freedom and computational expense. This will also make the operation and interpretation of the program easier to the user. The amplitude of motion in each normal mode of the blade in vacuum is treated as a generalized coordinate. Generalized coordinates $\mathbf{q}(t)$ are then related to the blade physical coordinates $\mathbf{u}(t)$ through the matrix equation:

$$\mathbf{q}(t) = \boldsymbol{\phi}^T \mathbf{u}(t) \quad (5)$$

where $\boldsymbol{\phi}$ is the $N \times m$ modal matrix of the blade. The above modal representation is used to write the dynamic response problem in terms of generalized coordinates as

$$\mathbf{M}_G \ddot{\mathbf{q}} + \mathbf{C}_G \dot{\mathbf{q}} + \mathbf{K}_G \mathbf{q} = \mathbf{Q}_A^r(\mathbf{q}(t), \dot{\mathbf{q}}(t), t) + \mathbf{Q}_A(t) + \mathbf{Q}_M(t) \quad (6)$$

where the matrices with subscript G correspond to the generalized coordinates contained in the vector $\mathbf{q}$. Each element of the vector $\mathbf{q}(t)$ measures the motion of the blade in the corresponding normal mode with the motion between adjacent blades differing only by a specified inter-blade phase angle. The matrices $\mathbf{M}_G$, $\mathbf{C}_G$ and $\mathbf{K}_G$ are given by

$$\begin{aligned} \mathbf{M}_G &= \boldsymbol{\phi}^T \mathbf{M} \boldsymbol{\phi} \\ \mathbf{C}_G &= \boldsymbol{\phi}^T \mathbf{C} \boldsymbol{\phi} \\ \mathbf{K}_G &= \boldsymbol{\phi}^T \mathbf{K} \boldsymbol{\phi} \end{aligned} \quad (7)$$

The generalized mass and stiffness matrices, $\mathbf{M}_G$ and $\mathbf{K}_G$, are diagonal because of the modal

transformation applied. The generalized damping matrix C_G is similarly diagonal because damping is assumed to be specified as modal damping. On the right hand side of eq. (6), the forcing functions Q_A^u , Q_A and Q_M are now expressed in terms of generalized coordinates and are given by

$$\begin{aligned} Q_A^u(q(t), \dot{q}(t), t) &= \phi^T F_A^u(u(t), \dot{u}(t), t) \\ Q_A(t) &= \phi^T F_A(t) \\ Q_M(t) &= \phi^T F_M(t) \end{aligned} \quad (8)$$

Since FREPS uses a linearized unsteady aerodynamic model, the equations of motion in generalized coordinates, given by eq. (6), are linear and lend themselves to solution in frequency domain, yielding significant savings in computational effort. The transformation to frequency domain is accomplished by making the standard assumption of harmonic excitation and motion:

$$\begin{aligned} Q_A(t) &= \bar{Q}_A e^{i\omega t} \\ Q_M(t) &= \bar{Q}_M e^{i\omega t} \\ q(t) &= \bar{q} e^{i\omega t} \end{aligned} \quad (9)$$

Thus, $\bar{Q}_A$ and $\bar{Q}_M$ are the components of the aerodynamic and mechanical excitation forces respectively at the frequency, ω . The linearity of the unsteady aerodynamic model also implies that the motion-dependent aerodynamic forces are proportional to the vibratory motion $u(t)$ of the blade and hence to $\bar{q}$, because of eqs. (5) and (9). This relationship can be written as

$$Q_A^u(q(t), \dot{q}(t), t) = A(\omega) \bar{q} e^{i\omega t} \quad (10)$$

where ω is the vibration frequency. The matrix A is of the order $m \times m$ and its ij -th element contains the generalized unsteady aerodynamic force in the i -th normal mode due to the vibration of the blade in the j -th natural mode at frequency ω . The ij -th element of A is calculated using appropriate interfaces between the finite element structural mesh and the unsteady aerodynamic computational mesh before and after the solution of the linearized unsteady potential equation using the j -th normal mode shape to specify the airfoil motion. We can now write the equations of motion in frequency domain as

$$[-\omega^2 M_G + i\omega C_G + K_G - A(\omega)] \bar{q} = \bar{Q}_A + \bar{Q}_M \quad (11)$$

Normalizing the natural mode shapes to unit generalized mass, and denoting the natural frequencies and the modal damping ratios of the blade vibration modes by ω_j , ζ_j , $j=1, \dots, m$

respectively, we obtain

$$\left[-\omega^2 \mathbf{I} + (1 + i2\zeta_j) [-\omega_j^2] - \mathbf{A}(\omega) \right] \bar{\mathbf{q}} = \bar{\mathbf{Q}}_A + \bar{\mathbf{Q}}_M \quad (12)$$

Eq. (12) is the basic equation solved by FREPS. Unsteady aerodynamic analysis provides $\mathbf{A}(\omega)$ and $\bar{\mathbf{Q}}_A$. Structural dynamic analysis provides $\mathbf{M}_G$, $\mathbf{C}_G$ and $\mathbf{K}_G$ or equivalently, ω_j , ζ_j and ϕ . A mechanical excitation model or data must provide $\bar{\mathbf{Q}}_M$ in the form of $\bar{\mathbf{F}}_M$.

The computational procedure to perform the aeroelastic analyses is schematically shown in Figure 1.4-1. The procedure involves reading in the blade geometric information, and the free-vibration eigenvalues and eigenvectors. After the blade geometry and modal information are checked, the aeroelastic strip definitions are ready for the analysis. Each strip is defined by specifying a locus of finite element nodal points, which coincide with the finite element configuration, along the airfoil surface. For each strip the inlet velocity triangle for the flow is specified. In addition, the aeroelastic parameters (e.g., frequencies and interblade phase angles of interest) are input.

Along each strip, the computational grids are generated for the calculation of the steady and unsteady potential flows. The method currently uses two different sets of grids for the steady and the unsteady flow problems. The steady flow solution for each strip is calculated and stored in database files on the computer. Similar information is initialized for the unsteady flow on each strip and stored in the database files.

FREPS begins the aeroelastic analysis by initializing the supplied database files which contain the steady potential solution and the computational grid information. The airfoil two-dimensional mode shapes are determined from the finite element eigenvectors for each strip. The mode shapes are interpolated from the finite element reference frame onto the unsteady aerodynamic computational grid. The unsteady flow due to the enforced airfoil mode shape is solved along each strip by a linearized unsteady potential approach described in reference [4]. Then the modal aerodynamic matrix, $\mathbf{A}$, is calculated from eq. (10). Refer to reference [3] for more information on the calculation of the $\mathbf{A}$ matrix.

The aerodynamic forces are integrated over the span and a complex eigenvalue problem of the form

$$\left[-\omega^2 \mathbf{I} + (1 + i2\zeta_j) [-\omega_j^2] - \mathbf{A}(\omega) \right] \bar{\mathbf{q}} = 0 \quad (13)$$

is solved for the flutter analysis option. This is done for all inter-blade phase angles specified in the input. All excitation forces are ignored. Note that eq. (13) is nonlinear in ω . The standard technique of using an assumed frequency ω_a for calculating the generalized aerodynamic

matrix and iterating to match the assumed frequency with the calculated frequency was employed. This technique transforms eq. (13) into a standard linear eigenvalue problem in ω^2 , which is readily solvable by standard techniques. For each of the eigenvalues, the aeroelastic damping μ and frequency ν are defined by

$$i\omega / \omega_a = \mu + i\nu \quad (14)$$

Instability or flutter is indicated by a positive value of μ . Frequency matching is attained when $\omega = \omega_a$. When aerodynamic damping is small, frequency matching is usually achieved by iteratively adjusting the value of ω_a until ν is approximately equal to unity. (For strict frequency matching, a subsequent iteration for zero damping must also be performed so that eq. (14) is valid in both real and imaginary parts [3].) Due to the large number of frequencies at which aerodynamic analyses are often required to be performed for frequency matching, an option is provided in FREPS for interpolation of aerodynamic coefficients. This interpolation procedure can sometimes be efficient, fast-running and reasonably accurate even though it is an approximation. The interpolation procedure can be used initially to estimate the matched frequency and, if necessary, subsequent runs can be performed without interpolation to improve frequency matching.

When the forced response option is selected, both the motion-dependent and the motion-independent aerodynamic forces as well as any user-specified external excitation forces are taken into account. Thus the response is calculated by inverting eq. (12) as

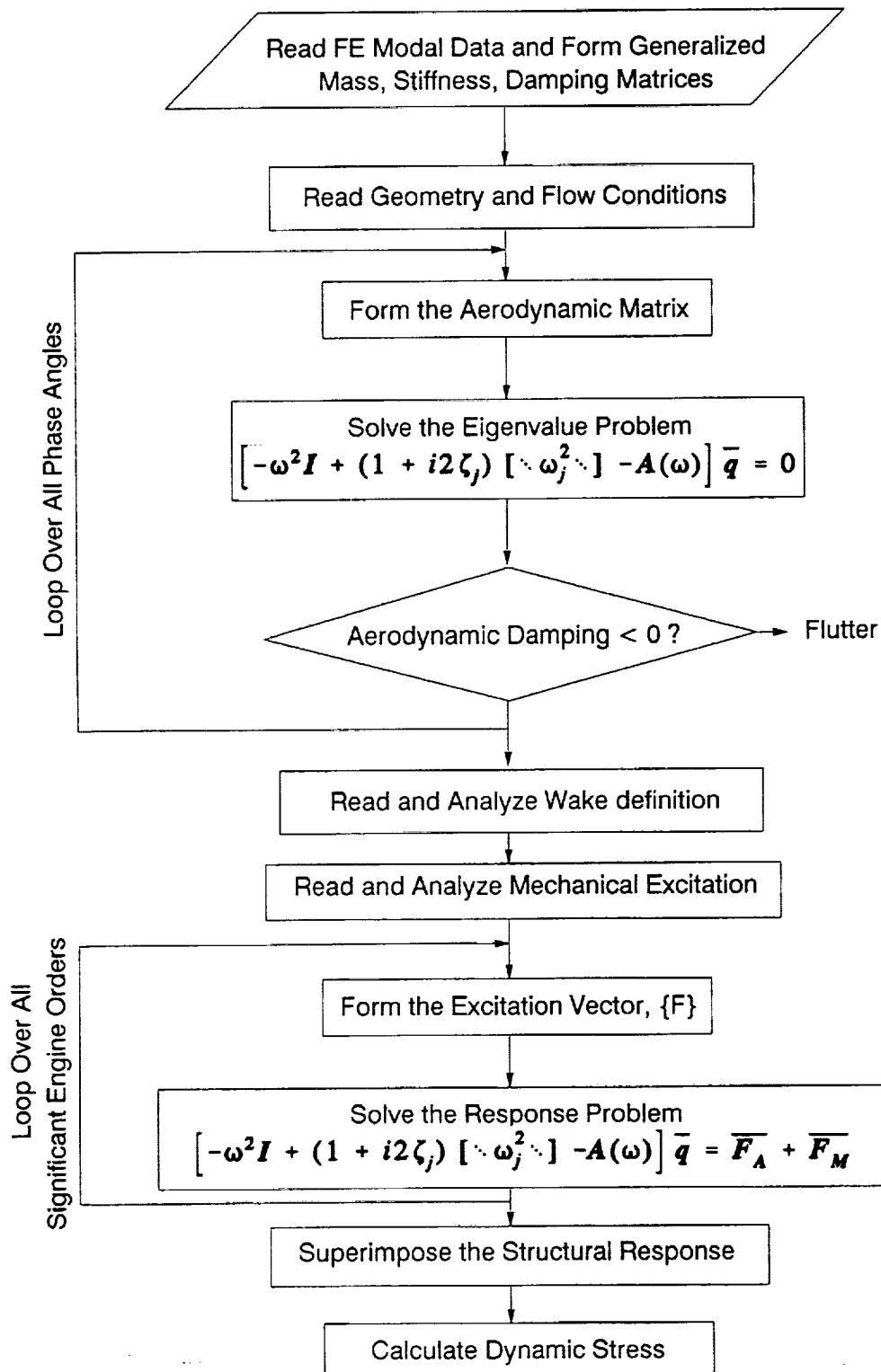
$$\bar{\mathbf{q}} = \left[-\omega_f^2 \mathbf{I} + (1 + i2\zeta_f) [-\omega_f^2] - \mathbf{A}(\omega_f) \right]^{-1} \{ \bar{\mathbf{Q}}_A + \bar{\mathbf{Q}}_M \} \quad (15)$$

Equation (15) is solved for the response $\bar{\mathbf{q}}$ in generalized coordinates after calculating all the required matrices and vectors. The generalized coordinate response is transformed to response in physical displacements (see eq. (5)) by

$$\bar{\mathbf{q}} = \boldsymbol{\phi}^T \bar{\mathbf{u}} \quad (16)$$

where $\bar{\mathbf{u}}$ is the normalized amplitude response in physical displacements. The stress response is obtained by similarly transforming the generalized coordinate response to physical stress response using the normalized modal stress vectors calculated by the finite element method.

Figure 1.4-1: FREPS Aeroelastic Stability and Forced Response Logic Flowchart



1.5 FREPS Geometric Conventions and Units

Due to the fact that the user must use three stand-alone computer codes before executing FREPS, it is very important to define/clarify the coordinate systems used by each system. In addition, the English Engineering System is used in the FREPS package and throughout this manual. The user may refer to Appendix A for the specific units that define the input values employed by FREPS.

The airfoil surface used in SFLOW and LINFLO is defined in an "airfoil coordinate system" wherein the leading edge and trailing edge Y coordinates are both zero. The airfoil is subsequently rotated up to the "cascade coordinate system" by a simple two-dimensional rotation and replicated tangentially to form the cascade (Figure 1.5-1). These two coordinate systems are very important when conducting the flow analysis. The airfoil shapes used in SFLOW and LINFLO are defined in the X-Y plane.

The airfoil strips generated from the finite element model must be in the X-Y plane, refer to Figure 1.5-2. The Z direction must define the height or span length of the model. Whereas the X direction defines the chordwise direction. This assumption is made in FREPS. Also, the finite element coordinate system and the airfoil coordinate system are one in the same.

Figure 1.5-1: Airfoil and Cascade Coordinate System Defined in SFLOW and LINFLO

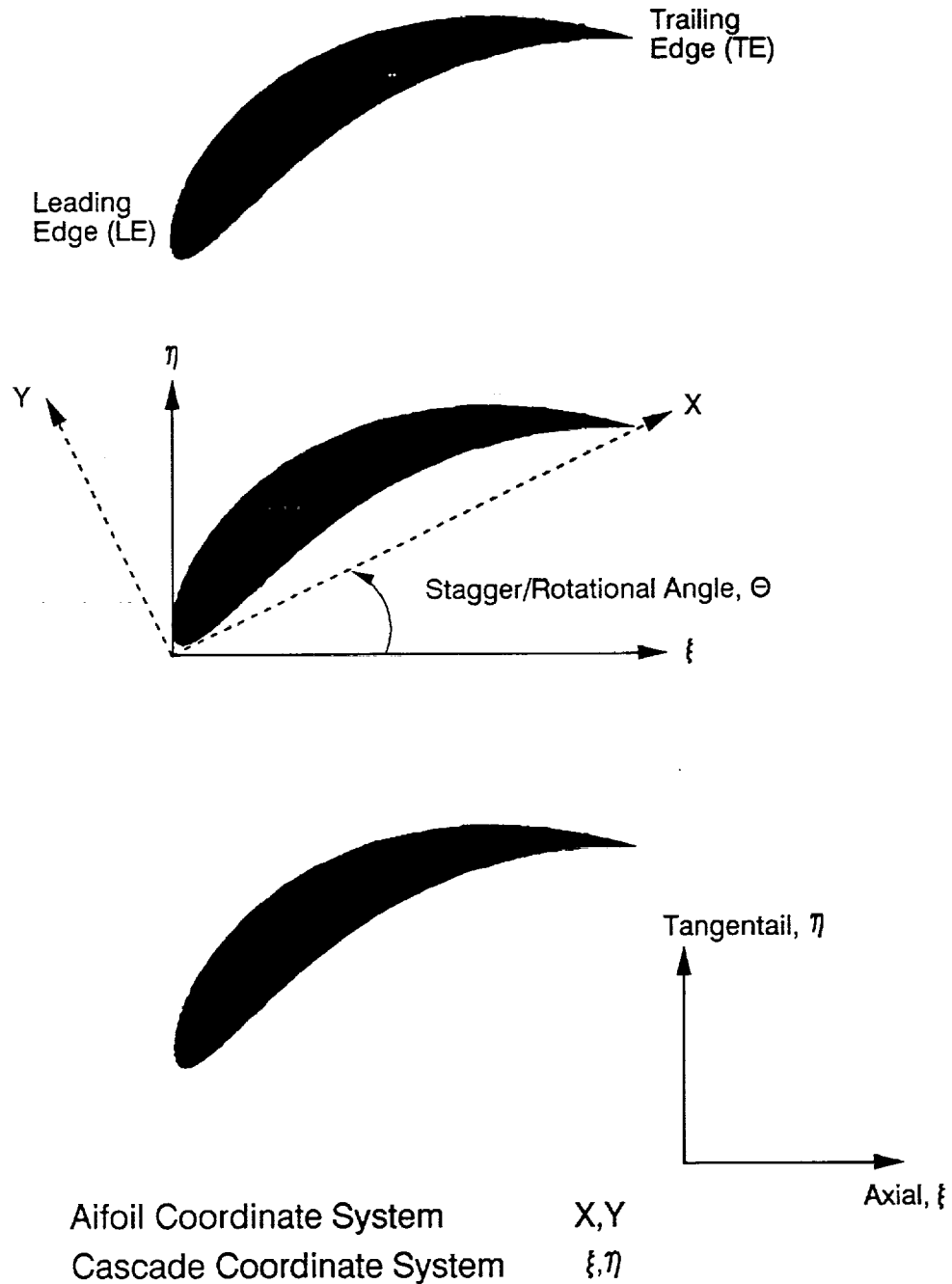
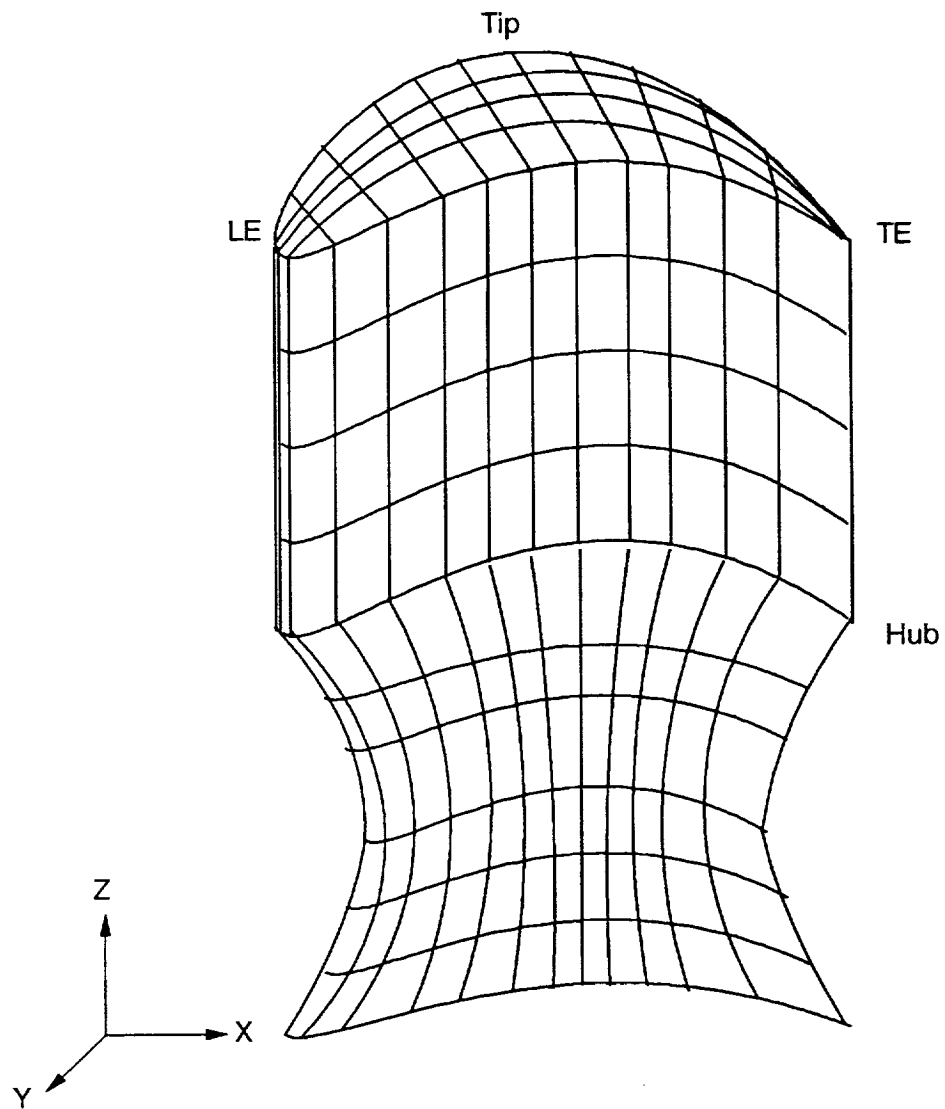


Figure 1.5-2: Finite Element Model Coordinate System Defined in MSC/NASTRAN



1.6 FREPS Input/Output File Structure

The file structure of the input/output (I/O) for FREPS is shown in Fig. 1.6-1. Four sets of input files are needed to execute the FREPS code:

- 1.) the primary input file
- 2.) the finite element model and modal results
- 3.) database files created from SFLOW
- 4.) the LINFLO generated database files

The number of database files from SFLOW and LINFLO depends on the number of strips analyzed. All strips designated in the primary FREPS input file must be assigned a database file each from SFLOW and LINFLO. Finally, FREPS produces the following output files:

- 1.) the standard output file
- 2.) extra output file for graphing
- 3.) PATRAN results files [7]
 - (a) mode shape from flutter analysis
 - (b) nodal displacements due to forced response excitation
 - (c) element principal stresses from excitation
- 4.) the calculated aerodynamic matrix

the contents of which the user can partially control using the "keyword" commands in the primary input file.

1.6.1 FREPS Input Files

The contents of the primary input file are shown in Fig. 1.6-2, also refer back to Table 1.2-2, pg. 1.2-5. The fluid flow within the turbomachinery blading is modeled along stacked two-dimensional stream-surface "strips" over the blade from the airfoil hub to tip (see Figure 1.2-2). Strips are defined by specifying nodes in the finite element model. Also, the aerodynamic, fluid, and thermodynamic properties are defined for the analysis. A description of the blade row, i.e., the number of blades, and radii at the blade hub and tip, is given along with the rotational speed, structural damping characteristics, mechanical excitation, flutter frequency and range. The user can request either flutter and forced response analysis.

FREPS is fully interfaced with MSC/NASTRAN in order for the structural geometry and free-vibration analysis of the turbomachinery component to be read in as input. The nodal coordinates from MSC/NASTRAN are needed to define the stream-surface strips assigned in the primary input file. Eigenvalues and eigenvectors are needed to couple the natural vibrational modes of the structure to the motion-dependent unsteady aerodynamic loads within the frequency

domain. In addition, modal stresses are used if a forced response analysis is requested. The user is referred to Appendix B for further information on the MSC/NASTRAN input and output files.

For all the strips defined in the primary input file, a binary database file each from SFLOW and LINFLO is required. Two different strips may be assigned the same database files from the aerodynamic codes. It is also noted that to create the LINFLO database files, the SFLOW database files are needed as input. These files contain the steady and unsteady aerodynamic results needed to calculate the aerodynamic loads acting on the component during vibration. Again, the user should review Appendices C and D for more information on SFLOW and LINFLO input and output.

1.6.2 FREPS Output File

The primary output file produced by FREPS includes two different groups of output, standard and optional output, shown in Table 1.6-1. The standard output group includes the echoing of the primary input file, a summary of the aeroelastic input, and the calculated results (e.g., aeroelastic eigensolution and dynamic stresses) used to evaluate the aeroelastic behavior. In addition, the code does have a primitive diagnostic capability that is printed to the primary output file, immediately following the echo of the input deck.

FREPS calculates the eigensolution for each phase angle requested and displays the results. The eigensolution uses the generalized stiffness and mass matrices and the modal aerodynamic matrix to determine the eigenvalues and eigenvectors. Both the real and imaginary parts are recorded. The dynamic stresses calculated during the forced response analysis due to the excitation are presented. For further information on the primary FREPS output file, the user should refer to Appendix E.

The second output group, labelled optional, allows the user to control the flow of output printed to the primary output file from within the FREPS input file. Some examples of the aeroelastic input are the strip airfoil coordinates, cascade properties, steady airfoil surface results, unsteady pressures along airfoil surface, unsteady nodal forces along surface, and strip unsteady aerodynamic analysis. The user, also, can invoke the plotting capabilities incorporated in FREPS via the IMSL routines. Line plots include the axial chord position versus the steady pressure, and the axial chord position vs. the unsteady pressure coefficient (real and imaginary).

A second output file can be generated which prints out critical data, shown in Table 1.6-2, in a tabular format to plot the data on an external graphics package. This file is created by stipulating the proper keyword in the FREPS input file. Data printed to the file includes:

- (1) airfoil cross-section contours in the airfoil and finite element coordinate systems for each strip in the FREPS input deck

- (2) axial chord length versus the steady static pressures
- (3) axial chord length versus unsteady pressure coefficients (real and imaginary)
- (4) interblade phase angles and corresponding eigenvalues (real and imaginary) for each mode specified

In order to evaluate the FREPS results, the FREPS package is interfaced with PATRAN, a pre and post processor. The user has the capability to generate PATRAN results files of the FREPS predictions by providing the proper keyword in the FREPS primary input deck. These files are used within PATRAN to graphically display the predicted results. Three types of results files can be created by FREPS for plotting on PATRAN. The first and second types are result quantities at the node points. For example, if a flutter analysis is performed the user can plot the mode shape. Whereas if a forced response analysis is chosen, the user can plot the displacements due to the loading. The third file contains the maximum principal stress for each element due to the excitation in the forced response analysis. These files are optional to the user. The format for these files are provided in Tables 1.6-3 and 1.6-4.

Figure 1.6-1: FREPS Input/Output File Structure

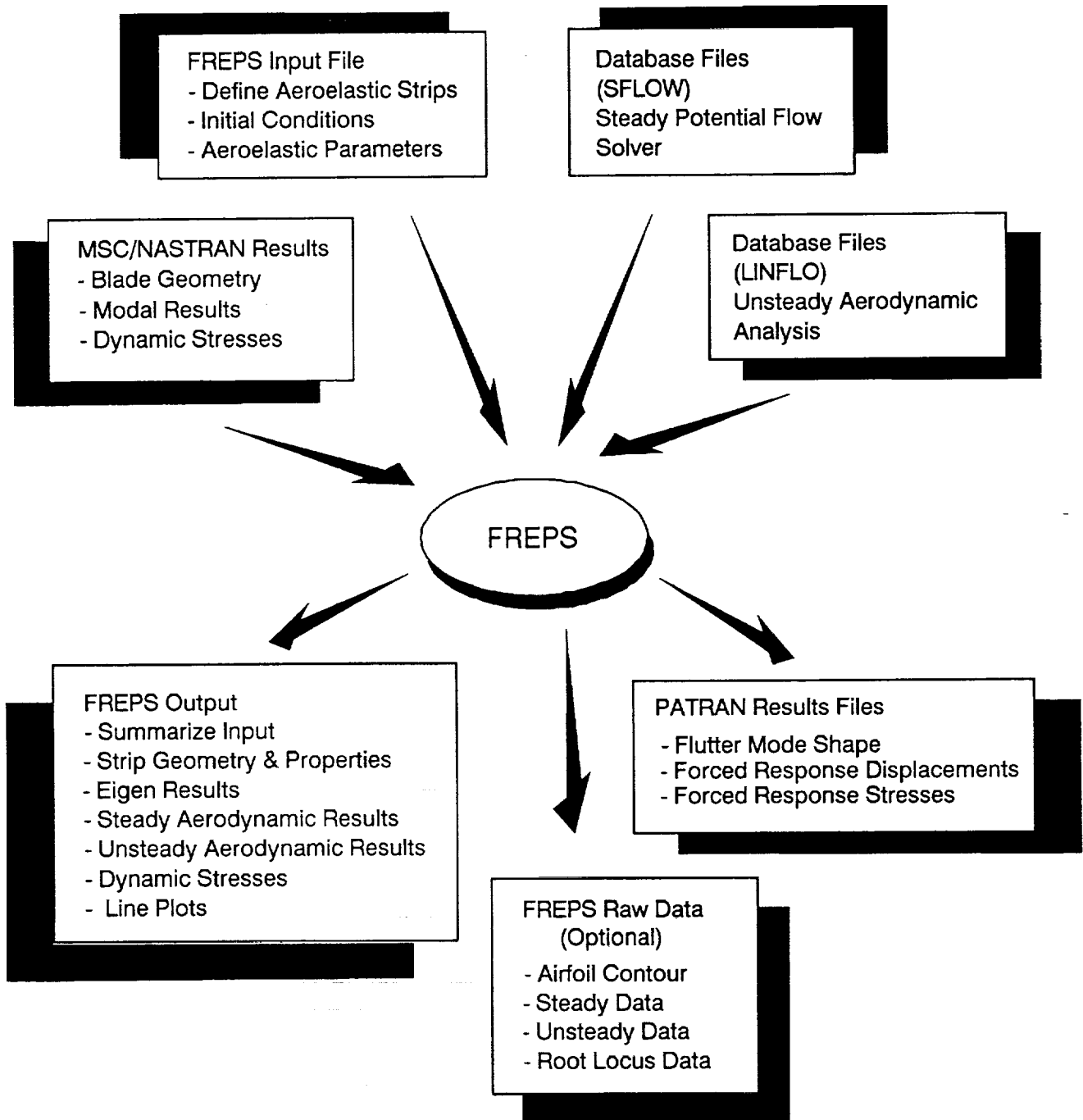


Figure 1.6-2: Summary of the FREPS Primary Input File

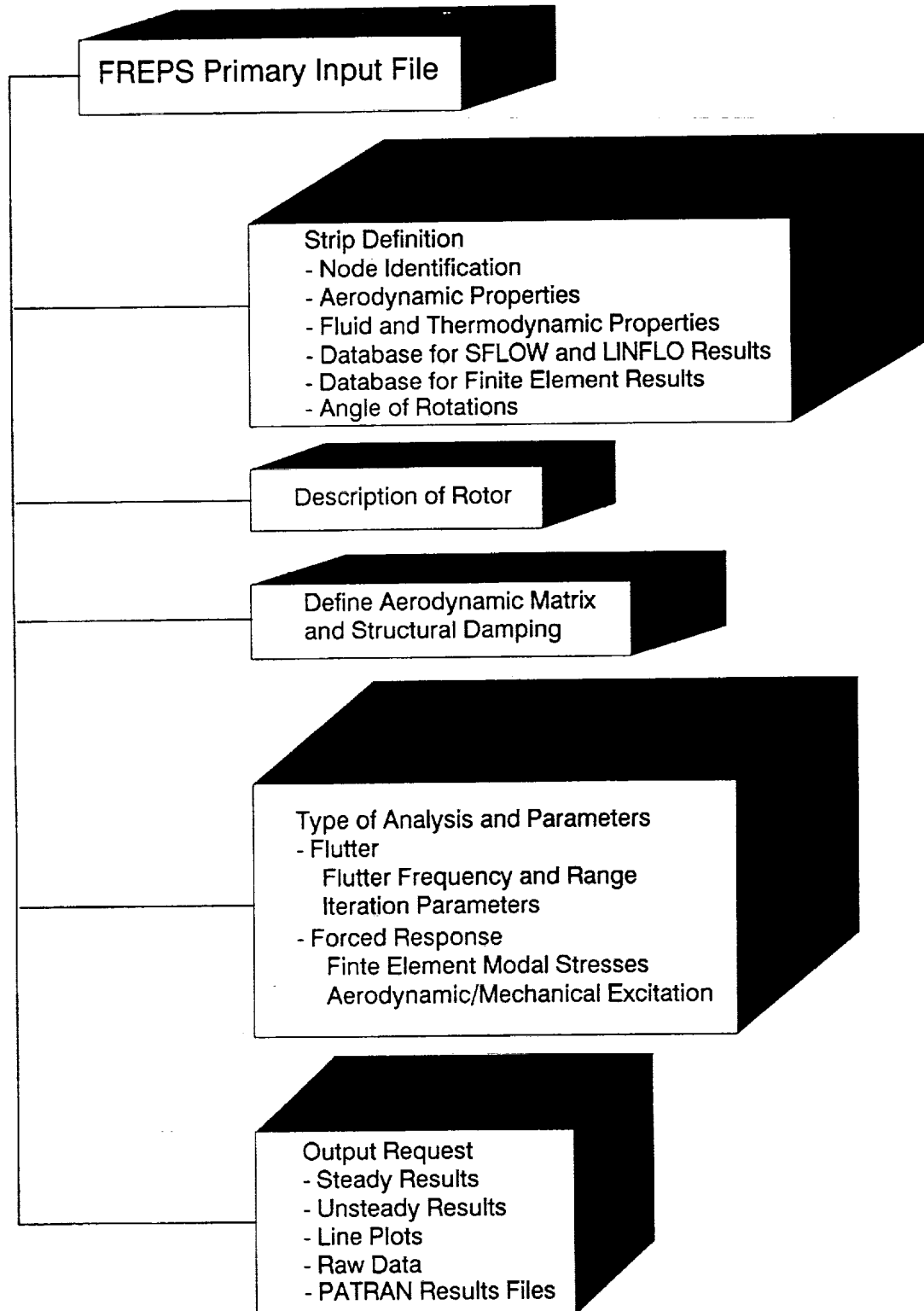


Table 1.6-1: Information Provided in the FREPS Primary Output File

Standard Output
- Echo of Input Data - Summary of Aeroelastic Input - Summary of Finite Element Results - Strip Cascade and Airfoil Properties and Geometry - Aeroelastic Eigensolution Results - Predicted Dynamic Stresses from Forced Excitation
Optional Output
- Strip Steady Aerodynamic Analysis - Strip Unsteady Aerodynamic Analysis - Calculated Modal Aerodynamic Matrix - Line Plots

Table 1.6-2: Data Furnished in the Extra FREPS Output File

Raw Data for Plotting Presented in a Tabular Format
- Airfoil Contour: X and Y coordinates of nodes that define the strips in the finite element coordinate system. - Normalized axial chord length and steady pressure. - Normalized axial chord length and unsteady pressure coefficients (real and imaginary). - Root Locus - for each specified interblade phase angle, the complex eigenvalues versus the real eigenvalue

Table 1.6-3: Nodal Results Data File for Flutter Mode Shape and Forced Response Displacements [7]*A set of records is created for both the real and imaginary parts*

<i>Record 1:</i>	TITLE	(80A1)
<i>Record 2:</i>	NNODES,MAXNOD,DEFMAX,NDMAX,NWIDTH	(2I5,E15.6,2I6)
<i>Record 3:</i>	SUBTITLE1	(80A1)
<i>Record 4:</i>	SUBTITLE2	(80A1)
<i>Record 5 to n+4:</i>	NODID,(DATA(j), j = 1,NWIDTH)	(I8,(5E13.7))

<i>Parameter</i>	<i>Description</i>
TITLE	Analysis description set internally, flutter or forced response (character)
SUBTITLE1	FREPS version number (character)
SUBTITLE2	Title supplied in FREPS primary input deck (character)
NNODES	Number of nodes (integer)
MAXNOD	Highest node ID number (integer)
DEFMAX	Maximum absolute displacement (real)
NDMAX	ID of node where maximum displacement occurs (integer)
NWIDTH	Number of columns after NODID for nodal information (integer)
NODID	Node ID number (integer)
DATA	Result quantities organized by column index (real or complex) $\Delta X, \Delta Y, \Delta Z$ with respect to the FE coordinate system

Table 1.6-4: Elemental Results Data File for Forced Response Stresses [7]

A set of records is created for both the real and imaginary parts

<i>Record 1:</i>	TITLE	(80A1)
<i>Record 2:</i>	NWIDTH	(I5)
<i>Record 3:</i>	SUBTITLE1	(80A1)
<i>Record 4:</i>	SUBTITLE2	(80A1)
<i>Record 5 to n+4:</i>	ID,NSHAPE,(DATA(j), j = 1,NWIDTH)	(2I8,/, (6E13.7))

<i>Parameter</i>	<i>Description</i>
TITLE	Analysis description set internally, flutter or forced response (character)
SUBTITLE1	FREPS version number (character)
SUBTITLE2	Title supplied in FREPS primary input deck (character)
NWIDTH	Number of columns of data stored in the file (integer)
ID	Element identification number (integer)
NSHAPE	Essential shape code (BAR = 2, TRI = 3, QUAD = 4, TET = 5, PYR = 6, WEDG = 7, HEX = 8; integer)
DATA	Result quantities organized by column index (real or complex) Column 1 - maximum principal stress for the element

FREPS INPUT PREPARATION

2.1 General Description

The MSC/NASTRAN system has served as a model for the FREPS user interface. The user prepares an input data file in which each line begins with a "keyword." Each keyword defines the data in a number of data fields that follow the keyword on the same line. Data may be placed anywhere within the boundaries of the appropriate fields. Keywords are used to supply numeric data as well as specify options that affect the execution of the program. The use of keywords allows the lines in the input data file to be arranged in any order that is meaningful to the user, although it is recommended that the organization of keywords in Table 2.1-1 be used as a guide to create the FREPS primary input file. For the user's convenience, the keywords described in Section 2.2 are given in alphabetical order.

It is recommended that particular attention be paid to Table 2.1-1. This table can be used to assist and guide the user in creating the FREPS primary input file. The table lists the general, mandatory, and optional keywords that are dependant upon other keywords to be included in the FREPS input file. This table will also help the user associate keywords that are dependant upon other keywords and eliminate unneeded keywords which could possibly cause other problems. Finally, the specification of the keywords has been designed for consistency among disciplines while allowing as much flexibility as possible in selecting run-time parameters. The following three run-time options are available in FREPS (see also Table 2.1-1):

1. **Flutter:** Flutter option (FLUTTER) is used to perform an aeroelastic stability analysis ignoring the excitation forces. Only the motion-dependent aerodynamic forces (and modal damping forces, if modal damping ratios are specified) are accounted for. This option essentially solves the free vibration problem.
2. **Steady Check-out:** This option (PERFORM) calculates the velocity triangles and steady pressure distributions around the blade for each aerodynamic strip. This option is basically used to check out the input file and database integrity and can be used along with the flutter or forced response options.
3. **Forced Response:** The forced response option (RESPONSE) is used to perform the analysis taking both the motion-dependent and motion-independent aerodynamic forces as well as mechanical excitation forces into account.

Table 2.1-1: Organization of Input File for FREPS

Keywords	Description	Page
----------	-------------	------

GENERAL

TITLE	Title printed on output.	2.2-39
\$	Comment symbol.	2.2-2

MANDATORY

Strip Definition

STRIP	Strip surface connectivity.	2.2-37
SFLUID	Strip fluid and thermodynamic properties.	2.2-32
SROTATE	Strip rotation angle definition.	2.2-35
SAERO	Strip aerodynamic properties	2.2-30
SGROUP	Strip group definitions.	2.2-34
AESET	Aeroelastic group set definition.	2.2-6
DATABASE	SFLOW and LINFLO database strip files.	2.2-11
FERESULT	Finite element database file of results.	2.2-15

Description of Blade Row

ROTOR	Blade row description and rotational speed.	2.2-29
-------	---	--------

OPTIONAL

Aerodynamic Matrix and Structural Damping

AMATRIX	Direct input of the aerodynamic matrix terms.	2.2-9
AMSAVE	Saves the calculated aerodynamic matrices.	2.2-10
SDAMP	Structural damping ratio definition.	2.2-31
MDAMP	Reserved for future use.	2.2-21

Keywords	Description	Page
----------	-------------	------

MANDATORY/OPTIONAL

At least one of the three analysis choices must be chosen.

Flutter Analysis & Parameters

FLUTTER	Request flutter analysis.	2.2-16
AEFREQ	Assumed flutter frequency for the calculation of [A].	2.2-3
AEFREQS	Frequency range of the assumed flutter frequency for the calculation of [A].	2.2-4
ITERATION	Iteration parameters for the frequency range in AEFREQS.	2.2-20
AEMODES	Define normal modes retained in analysis.	2.2-5
AESIGMA	Defines interblade phase angles for flutter analysis.	2.2-7

Performance Analysis

PERFORM	Request performance analysis.	2.2-26
---------	-------------------------------	--------

Forced Response Analysis & Parameters

RESPONSE	Request forced response analysis.	2.2-28
AEFREQ	Assumed flutter frequency for the calculation of [A].	2.2-3
AEFREQS	Frequency range of the assumed flutter frequency for the calculation of [A].	2.2-4
ITERATION	Iteration parameters for the frequency range in AEFREQS.	2.2-20
AEMODES	Define normal modes retained in analysis.	2.2-5
FREQUENCY	Forced response frequency range.	2.2-18
ELEMOUT	Specify the elements where principal stresses are calculated.	2.2-13
NODEOUT	Specify the node points where vibratory displacements are calculated.	2.2-22
STRFILE	Modal stress file definition.	2.2-36

Keywords	Description	Page
----------	-------------	------

Mechanical Excitation for Forced Response

FORCE	Specify concentrated harmonic mechanical excitation force.	2.2-17
GUST	Strip gust forcing function description.	2.2-19
AESIGMA	Defines interblade phase angles for gust forcing function.	2.2-7

OPTIONAL**Output Request**

DEBUG	Debug flag.	2.2-12
OUTPUT	Specific output requests.	2.2-23
PLOTOUT	Flag for line printer plots or raw data.	2.2-27
PATOUT	Generate PATRAN results file of FREPS results.	2.2-25

MANDATORY

END	Signals end of input data.	2.2-14
-----	----------------------------	--------

2.2 FREPS Keywords

This section describes the mnemonics needed to create the data deck for FREPS. The command name is at the top of the page followed by a description. The proper format and an example is also included. The character mnemonic must start in the first space and all pertinent data is to follow. All fields on an individual line can be separated by either a comma or blank space. A maximum of 80 spaces per line is used to signal the end of the line. Data is entered in as integer, real, or character specified in the field description table. For the real entries any valid real statement is acceptable.

The acronym "EOL" under the Field heading in the record definitions is equivalent to End Of Line. More specifically, the input data is entered in the remaining spaces up to the maximum of 80 spaces allowed per line. If additional values still exist after reaching the EOL/80 space maximum then continue on the next line with the appropriate mnemonic and syntax followed by the remaining values.

\$ Comment Symbol

Description:

The \$ record is used to make comments within the data deck. Also, the \$ can be used to inactivate the current line.

Format and Example:

1		80
\$ COMMENT		
\$ Any comment can be made, up to 79 characters.		

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	\$	character*1	Mnemonic name
2	COMMENT	character*79	User's comment

Notes:

- Comments can be inserted anywhere in the input data file except for the first line which is reserved for the TITLE record (refer to pg. 2.2-39).

AEFREQ Assumed Flutter Frequency

Description:

The AEFREQ record designates the assumed flutter frequency in the calculation of the aerodynamic matrix [A]. This is used in the stability, refer to keyword FLUTTER, and the forced response, refer to the keyword RESPONSE, analyses. To offer the user greater flexibility, two different ways are offered to assign the assumed flutter frequency: (1) use one of the natural frequencies by entering the mode number; or (2) give an approximation or exact value of the flutter frequency either in rad/sec or Hz.

Format and Example:

1		80
AEFREQ	MODE	f ω
AEFREQ		3500.0

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	AEFREQ	character	Mnemonic name
2	MODE	integer	Natural frequency mode
3	f	real	Flutter frequency (Hz)
4	ω	real	Flutter frequency (radian/second)

Notes:

1. Either a natural frequency mode, a f , or its equivalent ω is required for this record, the user has the flexibility to chose any one of the three ways.
2. The AEFREQ record should only appear once in the input deck, if two records exists the last appearance of the record has preference.
3. The natural frequency mode corresponds to the free-vibration results from MSC/NASTRAN.
4. In the above example, 3500.0 Hz is the assumed flutter frequency.
5. The AEFREQ record is not needed if the AEFREQS (pg. 2.2-4) and ITERATION (pg. 2.2-20) record are used.

AEFREQS Iterative Flutter Search Frequency Range**Description:**

The AEFREQS record describes a number of frequencies used to perform a cubic spline interpolation of the aerodynamic matrix. This keyword can be used in both the stability analysis, refer to the FLUTTER keyword (pg. 2.2-16) and forced response analysis, refer to RESPONSE keyword (pg. 2.2-28).

Format and Example:

1		80			
AEFREQS	f_1	f_2	f_3	...	f_n
AEFREQS	2000.0	3000.0	4000.0		

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	AEFREQS	character	Mnemonic name
2-EOL	f_i	real	Interpolation frequency

Notes:

1. The wider the frequency range, the more the number of frequencies that should be used.
2. Currently, a maximum of 20 interpolation frequencies can be used.
3. If the EOL (End Of Line) is reached before all the values are entered (*i.e.*, the user exceeds the 80 space per line limit), additional lines can be used. Each subsequent line should begin with the AEFREQS record followed by the remaining values.
4. For most cases, three frequencies should be used.
5. The ITERATION record, pg. 2.2-20, must be used with the AEFREQS record.
6. If the AEFREQ record is also present, then the AEFREQS record is ignored. In effect, the user should chose one or the other.

AEMODES Normal Modes Retained in Stability Analysis

Description:

The AEMODES record indicates the normal modes by number, as listed in the MSC/NASTRAN output file, retained in FREPS for the analysis. The numbering sequence from MSC/NASTRAN modal results are preserved in FREPS.

Format and Example:

1		80			
AEMODES	MODE ₁	MODE ₂	MODE ₃	...	MODE _n
AEMODES	1	2	3	4	

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	AEMODES	character	Mnemonic name
2-7	MODE _i	integer	Modal number

Notes:

1. A maximum number of six (n) natural modes can be identified.
2. The normal modes designated by the AEMODES record must also exist in the eigenvalue analysis performed by MSC/NASTRAN.
3. Any combination of the normal modes can be specified by the AEMODES record, e.g., 2, 3, 4 or 4, 2, 3.
4. This record is used in conjunction with the FLUTTER record (pg. 2.2-16) or RESPONSE record (pg.2.2-28), refer to Table 2.1-1, pg. 2.1-2, for other related keyword commands.
5. The first four normal modes are retained in the analysis in the above example.

AESET Aeroelastic Set Identifiers**Description:**

The AESET record activates the group identification number in the aeroelastic definition set. The group ID number is set through the SGROUP record (pg. 2.2-34).

Format and Example:

1		80
AESET	groupID ₁	groupID ₂ groupID ₃ ... groupID _m
AESET	100	200 300 400

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	AESET	character	Mnemonic name
2-EOL	groupID _i	integer	Group identification number

Notes:

1. Currently, a maximum of 80 group identification numbers can be used.
2. If the EOL (End Of Line) is reached before all the values are entered (*i.e.*, the user reaches the 80 space per line limit), additional lines can be used. Each subsequent line should begin with the AESET record followed by the remaining group identification numbers.
3. This record gives the user the flexibility to manipulate the property strip definitions very easily in the input deck.

AESIGMA Interblade Phase Angles

Description:

The AESIGMA record defines the interblade phase angles used in the aeroelastic analysis for the blades on the blade row. This keyword serves a dual purpose by listing the phase angles for the flutter analysis (FLUTTER, pg. 2.2-16) or mechanical excitations (GUST, pg. 2.2-19).

Format and Example:

1 80

AESIGMA	n_σ	σ_1	σ_2	...	σ_n	
AESIGMA	5	0.0	36.0	72.0	108.0	144.0
AESIGMA	5	180.0	216.0	252.0	288.0	324.0

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	AESIGMA	character	Mnemonic name
2	n_e	integer	Number of interblade phase angles submitted for the current record
3-EOL	σ_i	real	Interblade phase angles (°)

Notes:

1. For the FLUTTER option, the user has the choice of specifying a selected number of interblade phase angles (refer to above example) or using all phase angles that are allowed by cyclic symmetry. To use all possible interblade phase angles simply do not include this keyword in the input deck; the total number of allowable interblade phase angle angles is equal to the number of blades (N) on the blade row.
2. If used to define the gust loading (GUST), the AESIGMA card is mandatory.
3. The maximum number of interblade phase angles is currently 80.
4. The allowable interblade phase angles are calculated by the equation

$$\sigma_i = \frac{360^\circ}{N}(i - 1) \quad \text{where } i = 1 \text{ to } N$$

where N is the number of blades in the blade row (refer to the ROTOR record, pg. 2.2-29).

5. If the EOL (End Of Line) is reached before all the values are entered (*i.e.*, the user exceeds the 80 space per line limit), additional lines can be used. Each subsequent line should begin with the AESIGMA record followed by the number of phase angles being submitted and the remaining values.
6. In the example, two AESIGMA records are used to represent the phase angles requested by the user. It is assumed that 20 blades are located on the rotor. Every other interblade phase angle is used in the calculation, therefore, 5 angles are inserted on each line beginning with 0.0° and ending with 324.0°.

AMATRIX Input the Aerodynamic Matrix**Description:**

The AMATRIX record allows the user to directly input the aerodynamic matrix $[A]$, term by term. Each term is specified by position (row and column) followed by the real and imaginary part of the matrix.

Format and Example:

1					80
AMATRIX	row _i	col _j	Re(A) _{ij}	Im(A) _{ij}	
AMATRIX	2	3	4.6E06	-1.45E06	

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	AMATRIX	character	Mnemonic name
2	row_i	integer	Row number, i
3	col_j	integer	Column number, j
4	$Re(A)_{ij}$	real	Real part of A_{ij}
5	$Im(A)_{ij}$	real	Imaginary part of A_{ij}

Notes:

1. The maximum size of the aerodynamic matrix is 6x6.
2. If the user does not enter the entire matrix, FREPS will insert the appropriate terms as needed.
3. The size of the aerodynamic matrix for any given case is defined by the number of entries in the AEMODES record (pg. 2.2-5). For example, if the AEMODES record lists a total of 3 normal modes, then 9 possible entries for both the real and imaginary part, can be entered defining a 3x3 matrix.

AMSAVE Save Aerodynamic Matrices

Description:

The AMSAVE record saves the calculated aerodynamic matrix to a UNIX file. The user must specify the file name in both the FREPS primary input file and the job file (refer to pg. 3.3-1) if submitted as a batch job.

Format and Example:

1		80
AMSAVE	filename	
AMSAVE	A_NAME	

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	AMSAVE	character	Mnemonic name
2	filename	character*20	User specified file name

Notes:

1. Refer to Part III, pg. 3.1-1, for additional information on the I/O unit number and saving this file in a batch execution.
2. This keyword is optional, but the information contained in this file will be lost at the completion of execution.
3. In the above example, the aerodynamic matrix is saved in the file A_NAME.
4. The user must specify the file name in the job deck file, explained in Section 3.2, pg. 3.2-1.
5. An aerodynamic matrix is generated for each interblade phase angle, defined by the AESIGMA record (pg. 2.2-7) and the size of the matrix is specified by the AEMODES record (pg. 2.2-5). The total number entries, n , in the AEMODES record defines the size as a $n \times n$ matrix.

DATABASE Database File Names**Description:**

The DATABASE record defines the SFLOW [2] and LINFLO [3-4] database files that contain the steady and unsteady flow results. In addition, the corresponding strip is identified by defining the strip identification number.

Format and Example:

1		80
DATABASE	stripID	filename
DATABASE	12	BLADE12

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	DATABASE	character	Mnemonic name
2	stripID	integer	Strip identification number
3	filename	character*30	File name of database file

Notes:

- Each strip defined by the STRIP keyword, pg. 2.2-37, must have a SFLOW and LINFLO database file assigned to them. Multiple strips can have the same database files designated to them.
- The DATABASE strip identification number must match the strip identification number defined by STRIP record.
- Coordinate the filename with the job file deck created in Section 3.2 of Part III, pg 3.2-1, to make sure the filename is identified correctly. Both the SFLOW and LINFLO files should have the same prefix but different file extensions.
 - The SFLOW database files must have the file extension of U18.
 - The LINFLO database files must have the file extension of U17.
- Currently, the maximum number of DATABASE records accepted are 20.
- For the example, SFLOW and LINFLO database files exist under the filename BLADE12.U18 and BLADE12.U17, respectively. Also, a STRIP record exists with a strip identification number of 12.

DEBUG Debug Flag

Description:

The DEBUG record is a flag for de-activating the steady flow results located in the SFLOW database files.

Format and Example:

1		80
DEBUG	flag	nip
DEBUG	1	100

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	DEBUG	character	Mnemonic name
2	flag	integer	Debug flag 0 = activated 1 = de-activated
3	nip	integer	Number of intervals for numerical integration

Notes:

1. If this keyword does not exist in the input deck then the steady flow results are activated.
2. The interval points are used to integrate the unsteady pressure along each airfoil segment.
3. If the interval points are not entered the default value is 10.

ELEMOUT Elements for Principal Stress Calculation

Description:

The ELEMOUT record identifies the individual elements for which vibratory principal stresses are calculated. The elements must correspond to the finite element model used in the free-vibration analysis.

Format and Example:

1		80			
ELEMOUT	element ₁	element ₂	element ₃	...	element _m
ELEMOUT	111	112	113	211	210

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	ELEMOUT	character	Mnemonic name
2-EOL	element _i	integer	Element identification

Notes:

1. The maximum number elements that can be currently identified is 100.
2. This keyword is used when the forced response option is selected (RESPONSE, pg. 2.2-28).
3. If the EOL (End Of Line) is reached before all the values are entered (*i.e.*, the user exceeds the 80 space per line limit), additional lines can be used. Each subsequent line should begin with the ELEMOUT record followed by the remaining values.

END Signal End of Input Deck

Description:

The END record indicates the end of the input file has been reached. This should only be placed at the end of the input deck.

Format and Example:

1		80
END		
END		

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	END	character	Mnemonic name

Notes:

- Any input data following the END record is ignored and will not be executed.

FERESULT Finite Element Database File**Description:**

The FERESULT record defines the type of finite element output and the file name of the output results.

Format and Example:

1		80
FERESULT	FType	filename
FERESULT	NASTRAN	mscnast.out

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	FERESULT	character	Mnemonic name
2	FType	character*20	Finite element output type NASTRAN = MSC/NASTRAN
3	filename	character*20	UNIX file name of the output

Notes:

1. Currently, MSC/NASTRAN is the only FE solver supported in version 1.2 of FREPS.

FLUTTER Flutter Analysis

Description:

The FLUTTER record executes the stability analysis for the given finite element model and corresponding airfoil strips.

Format and Example:

1		80
FLUTTER		
FLUTTER		

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	FLUTTER	character	Mnemonic name

Notes:

- The following is a list of related commands that must exist to properly perform the stability analysis.

Keyword	
AEFREQ (2.2-3)	AEFREQS (2.2-4) ITERATION (2.2-20)
AEMODES (2.2-5)	
AESIGMA (2.2-7)	

- The FLUTTER option can not be used with the RESPONSE option (pg. 2.2-28).

FORCE Mechanical Harmonic Excitation Force

Description:

The FORCE record defines the magnitude of the concentrated harmonic mechanical excitation force in the forced response prediction (RESPONSE, pg. 2.2-28).

Format and Example:

1		80		
FORCE	node	cord	real _r	imag _r
FORCE	118	3	0.4	0.0

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	FORCE	character	Mnemonic name
2	node	integer	Node where force is applied
3	cord	integer	Direction of force applied at the current node 1 - X direction 2 - Y direction 3 - Z direction
4	real _r	real	Real part of force (lbf)
5	imag _r	real	Imaginary part of force (lbf)

Notes:

1. The maximum number of forces or FORCE records that can be applied is currently 80.
2. The direction of forces correspond to the global finite element coordinate system, as shown in Figure 1.5-2.

FREQUENCY Frequency Range

Description:

The FREQUENCY record defines the frequency range used in the forced response analysis (RESPONSE, pg. 2.2-28). The upper and lower limits along with the increment used to step through the range are established.

Format and Example:

1		80
FREQUENCY	f_{low}	f_{high} Δf
FREQUENCY	1000.0	4000.0 100.0

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	FREQUENCY	character	Mnemonic name
2	f_{low}	real	Start of frequency range (Hz)
3	f_{high}	real	End of frequency range (Hz)
4	Δf	real	Increment through the frequency range (Hz)

Notes:

1. The FREQUENCY keyword should only appear once in the primary input file.

GUST Gust Forcing Function

Description:

The GUST record defines the strip gust forcing function used in the forced response analysis (RESPONSE, pg. 2.2-28). The user specifies the strip identification number and magnitude of the gust. The interblade phase angle that the forcing function acts on is defined by the AESIGMA keyword (pg. 2.2-7).

Format and Example:

1		80
GUST	stripID	V _G
GUST	3	50.0

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	GUST	character	Mnemonic name
2	stripID	integer	Strip identification number
3	V _G	real	Velocity gust magnitude (ft/sec)

Notes:

1. The strip identification number to should match an existing strip defined by the STRIP keyword, pg. 2.2-37.

ITERATION Iteration Parameters for Flutter Search

Description:

The ITERATION record designates the parameters used for the iterative flutter search. This keyword must be accompanied by the AEFREQS record (pg. 2.2-4). The maximum number of iterations and the tolerance level are required to define the iteration parameters.

Format and Example:

1		80
ITERATION	nit	rtol
ITERATION	1000	1.0E-05

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	ITERATION	character	Mnemonic name
2	nit	integer	Maximum number of iterations
3	rtol	real	Relative tolerance for the eigenvalue

MDAMP Reserved for Future Use

Description:

The MDAMP keyword is reserved for future use.

Format and Example:

1 80

MDAMP
MDAMP

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	MDAMP	character	Mnemonic name

NODEOUT Nodal Points in Calculation of Displacements

Description:

The NODEOUT record specifies the node points for which vibratory displacements are calculated. This keyword is used when the forced response option (RESPONSE, pg. 2.2-28) is specified.

Format and Example:

1		80			
NODEOUT	node ₁	node ₂	node ₃	...	node _m
NODEOUT	114	117	118	116	125

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	NODEOUT	character	Mnemonic name
2-EOL	node _i	integer	Node identification

Notes:

1. The maximum number of nodes that can be specified is currently 100.
2. If the EOL (End Of Line) is reached before all the values are entered (*i.e.*, the user exceeds the 80 space per line limit), additional lines can be used. Each subsequent line should begin with the NODEOUT record followed by the remaining values.

OUTPUT Output Control**Description:**

The OUTPUT record provides the user with the flexibility to control the output supplied to the FREPS output file. This information is in addition to the standard output already furnished in the FREPS output file. A number of different options are supported.

Format and Example:

1		80			
OUTPUT	IP ₁	IP ₂	IP ₃	IP ₄	IP ₅
OUTPUT	0	0	1	0	

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	OUTPUT	character	Mnemonic name
2	IP ₁	integer	= 0 for no output (default) ≥ 1 for strip steady aerodynamic results and steady pressure distribution ≥ 2 for strip steady airfoil surface results
3	IP ₂	integer	= 0 for no output (default) ≥ 1 strip unsteady aerodynamic results and strip unsteady pressure distribution ≥ 2 unsteady nodal forces
4	IP ₃	integer	= 0 for no output (default) > 1 airfoil surface and mode shapes
5	IP ₄	integer	= 0 for no output (default) ≥ 1 system modal aerodynamic matrix, aerodynamic and mechanical generalized forces, and calculation CPU time for modal matrix
6	IP ₅	integer	= 0 for no output (default) = 1 flutter mode shape

Notes:

1. This card is optional in the input deck.
2. In Section 1.6 and Figure 1.6-3, pgs. 1.6-1 and 1.6-5, is the standard information provided in the FREPS output file.
3. In the example, no information is printed for the steady and unsteady results ($IP_1 = IP_2 = 0$) but the airfoil mode shapes are requested by $IP_3 = 1$.
4. Blanks to the right of the last entry are assumed to be zero, refer to the example, IP_5 although not entered would be equal to zero.

PATOUT PATRAN Results File**Description:**

The PATOUT record generates PATRAN [7] results files of the flutter mode shape and forced response displacements and stresses. After the keyword, the file type and file name are recorded.

Format and Example:

1		80
PATOUT	filetype	filename
PATOUT	STRESS	patran.stress

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	PATOUT	character	Mnemonic name
2	filetype	character*20	Type of output requested = EIGEN - flutter mode shape = DISP - forced response displacements = STRESS - forced response stresses
3	filename	character*20	User supplied UNIX file name

Notes:

1. This record can appear more than once in the FREPS primary input deck. For example, both the stress and displacement files can be generated in the same execution run.
2. Both the real and imaginary parts are printed in the results files. The real part is printed first and the corresponding imaginary part is second. This is repeated for each frequency in the range defined in the FREQUENCY record, pg. 2.2-18.
3. When plotting the stress results file in PATRAN, the maximum principal stresses are located in column 1.

PERFORM Performance Analysis

Description:

The PERFORM record executes a performance evaluation by calculating the velocity triangles and the steady pressure distributions around the blade for each of the aerodynamic strips. This option is used as a verification of the input file and database integrity.

Format and Example:

1 80

PERFORM
PERFORM

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	PERFORM	character	Mnemonic name

Notes:

1. This analysis option can be used along with the flutter or forced response analyses.

PLOTOUT Flag for Output**Description:**

The PLOTOUT record signals that either line printer plots are generated in the FREPS output file or the raw data is written to another output file to be utilized in an external plotting package. The user sets the data type after the keyword.

Format and Example:

1		80
PLOTOUT	data_type	filename
PLOTOUT	GENERIC	generic.data

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	PLOTOUT	character	Mnemonic name
2	data_type	character	Type of data: data_type = PRINTER then draw plots data_type = GENERIC generate data file
3	filename	character	If GENERIC = data_type then enter valid UNIX file name If PRINTER = data_type then leave blank

Notes:

1. The following data is plotted or stored: (1) the airfoil/strip contours; (2) the airfoil steady and unsteady pressure contours; and (3) root locus.
2. IMSL routines are used to generate the line plots (data_type = PRINTER) in the FREPS primary output file.
3. If the user chooses to create the additional output file and is running in batch mode, the generic file must be copied to the user's directory, refer to the FREPS job deck, pg. 3.2-1.
4. The additional generic file (data_type = GENERIC) contains output in a x-y column format for interfacing with external plotting packages of the users choice.

RESPONSE Forced Response Analysis

Description:

The RESPONSE record initiates the forced response analysis for the blade to either mechanical or aerodynamic forcing functions. The response is calculated in the modal coordinates.

Format and Example:

1		80
RESPONSE		
RESPONSE		

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	RESPONSE	character	Mnemonic name

Notes:

1. A punch file of the modal stresses from MSC/NASTRAN free-vibration analysis must be submitted as an input file through the STRFILE keyword (pg. 2.2-36) and must be identified in the job deck file (refer to Part III, section 3.2, pg. 3.3-1).
2. Refer back to Table 2.2-1 for related keywords that are used with the RESPONSE keyword.
3. The RESPONSE option can not be used with the FLUTTER option (pg. 2.2-16).

ROTOR Blade Row Characteristics

Description:

The ROTOR record describes the characteristics of the blade row; the number of blades, the speed, the radius of the tip and the radius of the hub.

Format and Example:

1					80
ROTOR	N	Ω	r_{hub}	r_{tip}	
ROTOR	20	10000.0	3.0	7.0	

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	ROTOR	character	Mnemonic name
2	N	integer	Number of blades in blade row
3	Ω	real	Blade row rotational speed (rpm)
4	r_{hub}	real	Radius at blade hub (in)
5	r_{tip}	real	Radius at blade tip (in)

Notes:

1. A maximum of 80 blades can currently be located on the blade row.
2. The input files for SFLOW and MSC/NASTRAN should be consistent in defining the same rotor speed defined in the FREPS primary input file.
3. The radii of the hub and tip are as shown in Figure 1.2-2 (pg. 1.2-7).
4. Only one ROTOR record is allowed in the FREPS input file.
5. The number of blades on the blade row is used for solidity and blade speed calculation.

SAERO Strip Aerodynamic Properties

Description:

The SAERO record defines the aero identification number and the aerodynamic properties for the strips.

Format and Example:

1		80			
SAERO	aeroID	W	β	M	H
SAERO	200	0.0	57.0	0.430	1.56

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	SAERO	character	Mnemonic name
2	aeroID	integer	Aero identification number
3	W	real	Inlet relative fluid velocity (fps)
4	β	real	Inlet relative flow angle w.r.t. axial (°)
5	M	real	Inlet relative Mach number
6	H	real	Strip aeroelastic stream-surface height (in.)

Notes:

1. The SAERO record is required for each strip.
2. The SGROUP keyword (pg. 2.2-34) assigns the aero properties to the individual strips.
3. For the relative fluid velocity and Mach number, only one of them is required. If both are entered, then the Mach number is used.
4. If the inlet flow angle and Mach number are unknown, both values can easily be found in the SFLOW output file.
5. The strip height is the radial distance (Z-coordinate) between strips, the strip and tip, or the strip and hub depending on the location of the strip. For example, if the strip is closest to the tip, the stream-surface height is equivalent to the distance from the tip to strip plus half the distance to the next strip.

SDAMP Structural Damping Ratio

Description:

The SDAMP record assigns the structural damping ratio to the modal coordinate to be used in the analysis.

Format and Example:

1		80
SDAMP	mode	ζ
SDAMP	2	0.034

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	SDAMP	character	Mnemonic name
2	mode	integer	Modal coordinate number
3	ζ	real	Modal structural damping ratio

Notes:

1. A maximum of six SDAMP records can be identified in the input file.
2. The assumption for modeling generalized damping is to assume equivalent modal damping applied to each individual vibrational mode in the structure.
3. The structural damping ratio correspond to each individual mode and when activated is used in calculating the diagonal terms of the generalized damping matrix.
4. If SDAMP record does not exist, then the damping is initialized to zero.

SFLUID**Strip Fluid and Thermodynamic Properties****Description:**

The SFLUID record specifies the fluid identification number and the fluid/thermodynamic properties used in the analysis.

Format and Example:

1							80
SFLUID	fluidID	T	p	ρ	a	γ	
SFLUID	100	518.69	14.69	0.0	1116.4	1.40	

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	SFLUID	character	Mnemonic name
2	fluidID	integer	Fluid identification number
3	T	real	Inlet static temperature (°R)
4	p	real	Inlet static pressure (psi)
5	ρ	real	Inlet fluid density (lbm/in ³)
6	a	real	Inlet sonic velocity (ft/s)
7	γ	real	Specific heat ratio

Notes:

1. A maximum of 20 SFLUID records can currently be used.
2. The SGROUP keyword, pg. 2.2-34, assigns the fluid identification number to the individual strips.
3. Only two of T, p, or ρ are required. The fluid is assumed to behave as an ideal gas and defined by the following equation:

$$p = T R \rho \quad (18)$$

where R is the gas constant. Also, the sonic velocity, a , can be calculated from the ideal gas relations by

$$a^2 = \gamma R T g \quad (19)$$

where g is gravity.

SGROUP Strip Group Identifier

Description:

The SGROUP record defines the relationships among the STRIP (pg. 2.2-37), SAERO (pg. 2.2-30), and SFLUID (pg. 2.2-32) records. The relationships are linked through the identification numbers assigned to each record. Each group of strip, aero, and fluid identification numbers are assigned a group identification number.

Format and Example:

1		80		
SGROUP	groupID	stripID	aeroID	fluidID
SGROUP	100	1	200	100

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	SGROUP	character	Mnemonic name
2	groupID	integer	Strip group identification number
3	stripID	integer	STRIP identification number
4	aeroID	integer	SAERO identification number
5	fluidID	integer	SFLUID identification number

Notes:

1. Currently, the maximum number of SGROUP records that can be used is 20.
2. The AESET keyword, pg. 2.2-6, activates the SGROUP record for the aeroelastic analysis.

SROTATE Strip Rotation Angle

Description:

The SROTATE record defines the alternate rotation angle between the finite element airfoil coordinates and the LINFLO airfoil coordinates.

Format and Example:

1		80
SROTATE	stripID	θ
SROTATE	1	57.0

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	SROTATE	character	Mnemonic name
2	stripID	integer	Strip identification number
3	θ	real	Rotation angle (°)

Notes:

1. A maximum of 20 SROTATE records can be currently used in the input file.
2. The user should refer to Section 1.5 (pg. 1.5-1) for the proper geometric convention used to determine the rotation angle.

STRFILE Modal Stress File

Description:

The STRFILE record defines the modal stress file generated by MSC/NASTRAN. This file is a punch file generated by NPRESS processing [1]. The file is only used when the RESPONSE record (pg. 2.2-28) is specified in the input deck.

Format and Example:

1 80

STRFILE	filename
STRFILE	strs.mscpunch

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	STRFILE	character	Mnemonic name
2	filename	character*30	UNIX file name

Notes:

1. This file must be defined in the job file deck, refer to Part III, pg. 3.1-1.
2. This file is generated by MSC/NASTRAN and the appropriate command should be inserted in the MSC/NASTRAN deck, refer to Appendix B and [1,7].

STRIP Strip Surface Node Connectivity

Description:

The STRIP record defines the outer airfoil shape from the finite element model by defining the nodal connectivity. Strips should be chosen carefully from the finite element model by selecting the strips in the most critical areas interpreted from the mode shapes.

Format and Example:

1		80
STRIP	stripID	node ₁ node ₂ ... node _m
STRIP	1	235 236 237 238 239 240 241 242 243 244 245
STRIP	1	251 252 253 254 256 257 258 259 260 270 271
STRIP	1	207 208 209 210 211 212 213 234 235

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	STRIP	character	Mnemonic name
2	stripID	integer	Strip identification number
3-EOL	node _i	integer	Node point identification number

Notes:

1. Currently, at least one strip must be defined and a maximum of 20 strips can be used.
2. At the present, a maximum of 80 nodes can be used to define the outer shell of the airfoil, no interior nodes are to be listed.
3. Additional STRIP records are used consecutively if all nodes describing the airfoil do not fit on one line. If this is the case, then after the keyword, the same strip identification number is used and the remaining nodes are entered, as demonstrated in the above example.
4. The nodal coordinates outlining the airfoil are assumed to be in the X-Y plane of

- the finite element model.
5. First and last grid must be identical.
 6. The trailing edge (TE) is assumed to have the greatest X-coordinate (Figure 1.5-2).
 7. The node numbering must start at the trailing edge upper surface (suction-side) and continue in a counter-clock-wise direction to the lower surface (pressure-side) of the trailing edge.
 8. Refer to Fig. 1.2-1 for the direction of the numbering sequence.
 9. For consistency make sure the airfoil described in the SFLOW input and the FREPS input files are entered in this manner.

TITLE Title

Description:

The TITLE record gives the user the opportunity to enter a description of the model or problem. The title description is used as a header description in the output file.

Format and Example:

1		80
TITLE description		
TITLE Any title can be typed in this space.		

Field Description:

<u>Field</u>	<u>Name</u>	<u>Type</u>	<u>Description</u>
1	TITLE	character	Mnemonic name
2	description	character*74	Descriptive title

Notes:

1. The TITLE record and descriptive title can not exceed 80 characters and a space must exist after the TITLE keyword.
2. The TITLE record must be the first record in the input deck.
3. Use only one TITLE keyword throughout the input deck.

2.3 Sample Input Data File

To demonstrate the forced response capabilities of FREPS, an example input data file is presented to aid the user in preparing their own input decks. The turbine blade illustrated in Fig. 1.3-1 (Section 1.3, pg. 1.3-10) is used to predict the vibratory response for an assumed gust excitation. The preliminary steps outlined in Table 1.2-1 are assumed to have been completed by the user. In addition, the input files received from the steady aerodynamic, unsteady aerodynamic, and finite element eigen analyses, already generated for the first example, can be utilized for this case. Their file names are:

- 1.) mscnastran.output
- 2.) airfl_1.U18, airfl_2.U18, airfl_3.U18, airfl_4.U18
- 3.) airfl_1.U17, airfl_2.U17, airfl_3.U17, airfl_4.U17

The first part of the FREPS input file, refer to Table 2.3-1, is similar to the previous example case and will only be briefly explained in this section. Four radial strips were chosen to represent the blade model in FREPS. For each strip, 53 nodes define the airfoil shell from the finite element model. In Tables 1.3-3 and 1.3-4 (pg. 1.3-9) are the fluid properties and aeroelastic strip properties. Finally, the strip database files containing the SFLOW and LINFLO results are defined followed by the finite element database of the structural dynamic results. The user does not have to rerun the steady and unsteady flow analyses or finite element analysis that were used for the flutter analysis presented earlier.

Fifty blades are on the rotor operating at 6000 rpm. The hub radius is 0.5 in. and the tip radius is 1.5 in. All strips defined in the input are used, therefore, the AESET keyword activates all strips by defining their group identification numbers. The first four natural frequency modes are used in the forced response analysis. This concludes the input that is identical to the first example case.

Since this blade is only a test case, the gust amplitude and phase angle both are assumed. In this case, the gust is presumed to be a normal gust with an inlet velocity of 300-120 in/sec from tip to hub and an interblade phase angle equivalent to 72.0° . As a result, the AESIGMA record specifies one interblade phase angle (72.0°) at the phase where the excitation is applied. The GUST record defines the strip by the strip identification number and the amplitude of the gust excitation.

Following the GUST records in the input deck is the FREQUENCY record which defines the upper and lower bounds of the frequency range. Also, an increment value is given to step through the frequency range. The NODEOUT and ELEMOUT records list specific nodes and elements where the vibratory displacements and the principal stresses are requested. All the displacement and stress results defined in the input deck are in the tip region of the blade. The NODEOUT record states five nodes where vibratory displacements are requested. Likewise,

the ELEMOUT record displays four elements were the principal stress are calculated.

A punch file of the stresses (received from MSC/NASTRAN) from the eigenvalue calculations must be included in the input deck for forced response analysis. The STRFILE record indicates the file name and a non-zero dummy integer to define the I/O unit number. The non-zero integer is also a flag to indicate that a modal stress file from MSC/NASTRAN must be read. Along with the forced response analysis, indicated by the RESPONSE record, the performance analysis record, PERFORM, is also included to check-out the input and database integrity. This option calculates the velocity triangle and steady pressure distributions around the blade for each strip. Since some assumptions are made for this test case, the performance analysis will help verify the integrity of the blade analysis from SFLOW and LINFLO.

Table 2.3-1: FREPS Sample Input

```

1 . . . . . 80
TITLE  TURBINE BLADE -- FORCED RESPONSE ANALYSIS
$=====
$          TURBINE BLADE AEROELASTIC ANALYSIS
$ ANALYZES THE BLADE USING 4 RADIAL STRIPS FROM THE BLADE, GUST LOADING
$=====
$
$          STRIP DEFINITION
$
$          closest to tip
STRIP  1      3      27      75      159      243      327      411
STRIP  1      495     579     663     747     831     915     999
STRIP  1     1083    1206    1321    1394    1397    1401    1405
STRIP  1     1409    1413    1417    1421    1424    1427    1428
STRIP  1     1426    1422    1419    1414    1410    1406    1402
STRIP  1     1398    1318    1203    1080     996     912     828
STRIP  1       744     660     576     492     408     324     240
STRIP  1       156      72      26       3
$
STRIP  2       8       37      95     179     263     347     431
STRIP  2      515     599     683     767     851     935    1019
STRIP  2     1103    1226    1341    1448    1547    1665    1764
STRIP  2     1767    1853    1897    1901    1904    1907    1908
STRIP  2     1906    1902    1899    1894    1850    1662    1544
STRIP  2     1445    1338    1223    1100    1016     932     848
STRIP  2       764     680     596     512     428     344     260
STRIP  2       176      92      36       8
$
STRIP  3       13      47     115     199     283     367     451
STRIP  3      535     619     703     787     871     955    1039
STRIP  3     1123    1246    1361    1468    1567    1685    1793
STRIP  3     1878    1951    2003    2036    2053    2058    2052
STRIP  3     2030    1993    1934    1867    1784    1682    1564
STRIP  3     1465    1358    1243    1120    1036     952     868
STRIP  3       784     700     616     532     448     364     280
STRIP  3       196     112      46      13
$
$          closest to hub
STRIP  4       18      60     135     219     303     387     471
STRIP  4      555     639     723     807     891     975    1059
STRIP  4     1143    1266    1381    1492    1593    1711    1810
STRIP  4     1893    1912    1916    1919    1922    1923    1921
STRIP  4     1917    1914    1909    1872    1789    1690    1575
STRIP  4     1479    1375    1263    1140    1056     972     888
STRIP  4       804     720     636     552     468     384     300
STRIP  4       216     132      56      18
$
$          FLUID AND AERO DESCRIPTION FOR STRIPS
$
$          fluidID      T(°R)      p(psi)      ρ(lbm/in³)      a(fps)      γ
SFLUID      100      518.69,  14.69,      0.0000,      1116.4,      1.4
$
$          stripID      θ
SROTATE      1          0.0
SROTATE      2          0.0
SROTATE      3          0.0
SROTATE      4          0.0
$

```

2.3 Sample Input Data File

```

$      aeroID      W       $\beta$       M      H
SAERO    100,      0.0,    114.0,    0.3800,    0.225
SAERO    200,      0.0,     63.0,    0.3850,    0.250
SAERO    300,      0.0,     57.0,    0.4300,    0.250
SAERO    400,      0.0,     55.0,    0.4500,    0.775
$
$      groupID    stripID    aeroID    fluidID
SGROUP    100         1         100         100
SGROUP    200         2         200         100
SGROUP    300         3         300         100
SGROUP    400         4         400         100
$
$      STRIP DATABASE DEFINITIONS
$
$      stripID    filename
DATABASE    1      airfl_1
DATABASE    2      airfl_2
DATABASE    3      airfl_3
DATABASE    4      airfl_4
$
$      FINITE ELEMENT DATABASE RESULTS
$
$      filename
FERESULT    mscnastran.output
$
$      ROTOR DESCRIPTION
$
$      NofBLADES       $\Omega$ (rpm)      hubRADIUS      tipRADIUS
ROTOR        50          6000.0         0.5          1.5
$
$      AEROELASTIC SET DEFINITIONS
$
$      groupIDs
AASET        100    200    300    400
$
$      INTERBLADE PHASE ANGLE OF GUST LOAD
$
AESIGMA      1    72.0
$
$      GUST LOADS
$
GUST        1         25.0
GUST        2         20.0
GUST        3         15.0
GUST        4         10.0
$
FREQUENCY    9700.0    10100.0    20.00
$
NODEOUT      7851      7860
ELEMOUT      2886      2917      2892
$
$      MSC/NASTRAN PUNCH FILE OF STRESSES
$
STRFILE      stress.punch.file
$
$      PERFORMANCE AND FORCED RESPONSE ANALYSES REQUESTED
$
PERFORM
RESPONSE
$
END

```

PROCEDURE TO RUN FREPS

3.1 Introduction

The FREPS computer code has been setup and executed on the Cray Y-MP computer at NASA Lewis Research Center. The Cray Y-MP uses the UNICOS operating system. Located in Table 3.1-1 are the unit numbers for the input/output files required by FREPS. All unit numbers are set internally by the system. The FREPS primary input filename must be named "aeinput." To meet this criteria the user has two options when naming this file: (1) just name their file aeinput; or (2) name the file to the user's specifications and rename it in the job file (if running in batch mode).

The SFLOW and LINFLO database files must have the file extension of "U18" and "U17," respectively. Their file names must be the same, *e.g.*, for a given strip, the SFLOW and LINFLO database file names are strip1.U18 and strip1.U17. The file names for the database files are selected by the user and are defined in the primary input file by the "DATABASE" record command, refer to pg. 2.2-11 of Section 2.2. Similarly, the structural dynamic results are defined in the input deck by the "FERESULT" record with the file name also given. It is important to remember that the UNICOS operating system is case sensitive when defining file names and types.

The MSC/NASTRAN stress punch file is optional, but if used the filename is activated by the "STRFILE" command in the primary input deck, again refer to Section 2.2 pg. 2.2-36. The raw data, aerodynamic matrix, and PATRAN results files are only saved when the user indicates in the primary FREPS input file (*i.e.*, the proper keyword is listed in the input deck) that these files are to be generated. In addition, the files must be copied to the user's directory if executed in batch mode, which is discussed further in the next section.

Additional read/write files used by FREPS are located in Table 3.1-2, the user does not have to assign any filenames to them. The files are deleted at the completion of the FREPS execution.

Table 3.1-1: Input/Output Units for FREPS

Description	Input	Output	File Name
FREPS input	10		aeinput
MSC/NASTRAN output file	13		supplied by user
MSC/NASTRAN stress punch file	12		supplied by user
LINFLO files	17		*.U17
SFLOW database files	18		*.U18
FREPS output file		6	supplied by user
FREPS raw data		14	supplied by user
PATRAN results file EIGEN DISP STRESS		26 27 28	supplied by user
Storage of aerodynamic matrix		20	supplied by user

Table 3.1-2: Additional Read/Write Files Used by FREPS, Hidden to the User

Description	Unit Number
SCRATCH file	11
SCRATCH file	15
Storage of general blade displacements	16
SCRATCH file	23
SCRATCH file	24
SCRATCH file	25
SCRATCH file	30
Used by READSD as scratch file	31

3.2 Run FREPS on the Cray Y-MP

The job file to execute FREPS on the Cray Y-MP is submitted while interactively on the Cray Y-MP computer system. The potential user is advised to use the work directory made available to them on the LeRC Cray Y-MP due to the enormous space requirements for the SFLOW and LINFLO database files. An example listing of a job file submitted from the Cray Y-MP in batch mode is given as follows:

```
# USER=userid PW=password                # valid Cray userid and password
# QSUB -r frepsjob                        # jobname
# QSUB -lT 1200                           # cpu time limit
# QSUB -lM 4.0Mw                          # memory size limit
# QSUB -eo                                # combine stderr and stdout
# QSUB -o /wrk/freps.out.put             # output file named
ja                                         # job accounting data
set -kx                                  # show commands in batch output
#
cp /wrk/example/aeinput .                # FREPS primary input file
cp /wrk/example/mscnastran.output .      # MSC/NASTRAN output file
cp /wrk/example/dbase*.U* .              # SFLOW and LINFLO database files
cp /wrk/example/stress .                 # MSC/NASTRAN punch file
/wrk/freps/freps.x                       # Execute the FREPS code
debug -B -m 2 -s /wrk/freps/freps.x -V   # Debug option
flowview -lmt
cp generic.data /wrk/example/raw.data    # Copy raw data file to directory
cp A_MATRIX /wrk/example/a.matrix        # Copy aerodynamic modal matrix
cp ptrn.strs /wrk/example/patran.stress  # Copy PATRAN results file
ja -clsfh
rm -f .jacct*
```

In the above job file, the FREPS primary input file is labeled **aeinput**, the MSC/NASTRAN output is named **mscnastran.output**, and the database files are named **dbase1.U17** and **dbase1.U18** but are represented in the job file using the wildcard symbol (*). In addition, the modal stresses from MSC/NASTRAN are stored in the file **stress**. All of the input files must reside in the **example** subdirectory in the **wrk** directory and the executable file (**freps.x**) for FREPS is located in the **freps** subdirectory in the **wrk** directory. A CPU time of 1200 seconds is recommend for most jobs.

The files **generic.data**, **A_MATRIX**, and **ptrn.strs** are copied to the user's directory by the UNIX copy command **cp** and named **raw.data**, **a.matrix**, and **patran.stress**. The files must be copied in this fashion if the user is to recover the information inside the files. This file is only required when FREPS is executed in the batch mode. All three of these files must be defined in the FREPS primary input file by their appropriate keywords, i.e., **PLOTOUT** designates FREPS to generate **generic.data**, **AMSAVE** instructs FREPS to save the aerodynamic modal matrix to **A_MATRIX**, and **PATOUT** generates the PATRAN results file, **ptrn.strs**. Likewise, the **DATABASE** and **STRFILE** keywords assigns the database file names and NASTRAN stress file that are stipulated in the FREPS job deck.

REFERENCES

1. McCormick, C.W., ed.: MSC/NASTRAN User's Manual, Vol. I and II, MacNeal-Schwendler, 1983.
2. Hoynaik, D., and Verdon, J.M., "Development of a Steady Potential Solver for Use With Linearized, Unsteady Aerodynamic Analyses," *NASA TM 105288*, September, 1991.
3. Verdon, J.M., and Caspar, J.R., "Development of a Linear Unsteady Aerodynamic Analysis for Finite-Deflection Subsonic Cascades," *AIAA Journal*, Vol. 20, pp. 1259-1267.
4. Verdon, J.M., and Caspar, J.R., "A Linearized Unsteady Aerodynamic Analysis for Transonic Cascades," *Journal of Fluid Mechanics*, Vol. 149, pp. 403-429.
5. Smith, T.E., "A Modal Aeroelastic Analysis Scheme for Turbomachinery Blading," *NASA CR 187089*, March, 1991.
6. Murthy, D. V. and Pierre, C., "Stochastic Sensitivity Measure for Mistuned High-Performance Turbines", *NASA TM-105821 (ICOMP-92-13)*, August 1992.
7. PATRAN II Users Manual, PDA Engineering, Santa Ana, CA 1984.
8. Lawrence, C., Aiello, R.A., Ernst, M.A., and McGee, O.G., "A NASTRAN Primer for the Analysis of Rotating Flexible Blades," *NASA TM 89861*, 1987.
9. Schaeffer, H.G., MSC/NASTRAN Primer Static and Normal Modes Analysis, MacNeal-Schwendler, 1979.
10. Verdon, J.M., Hall, K.C., "Development of a Linearized Unsteady Aerodynamic Analysis for Cascade Gust Response Predictions," *NASA CR 4308*, 1990.

APPENDIX A

Nomenclature

Symbols

A	Modal unsteady aerodynamic matrix
a	Fluid sonic velocity (feet/second)
C	Damping matrix
$\overline{F}_A$	External aerodynamic forcing function
$\overline{F}_M$	External mechanical forcing function
f	Flutter frequency (hertz)
H	Strip stream surface height (inch)
I	Unity matrix
i	$\sqrt{-1}$
K	Stiffness matrix
M	Mach number
M	Mass matrix
m	Number of degree-of-freedom per blade
N	Number of blades in the blade row
n	Number of modes used in flutter analysis
p	Static pressure (psi)
Q	Forcing function in generalized coordinates
$\overline{q}$	Modal coordinates

R	Gas constant
r	Radius (inch)
T	Temperature (° Rankine)
t	Time step
u	Motion of blade
V _G	Gust velocity (feet/second)
W	Fluid velocity (feet/second)
X,Y,Z	Finite element model and airfoil coordinate system
1,2,3	Finite element model and airfoil coordinate system

Greek

β	Relative flow angle (degree, °)
β_r	Constant interblade phase angle (°)
γ	Gas specific heat ratio
ζ	Structural damping ratio
θ	Rotational angle or stagger angle (degree, °)
π	Pi
ρ	Density (lbm/in ³)
σ	Interblade phase angle (degree, °)
ϕ	Modal coordinates of blade, size Nxm
Ω	Rotational speed (rpm)
ω	Flutter frequency (radian/sec)
ξ, η	Cascade coordinate system

APPENDIX B

MSC/NASTRAN

To avoid the development of an independent and expensive structural dynamic analysis program, FREPS structural dynamic model was developed to be easily interfaced with the normal modes procedure of MSC/NASTRAN [1]. Thus, the full power of all the capabilities of MSC/NASTRAN in modeling and analysis can be used with the aeroelastic and aerodynamic analysis capabilities of FREPS. MSC/NASTRAN was selected for this purpose because of its wide availability, established successful correlations with test experience and proven computational efficiency. The list below contains the required information (by FREPS) produced by the MSC/NASTRAN analysis.

- Standard MSC/NASTRAN Output:

- (1) Grid (nodes) identification and corresponding coordinates to define the strip locus.
- (2) Elements (3-dimensional) and their connectivity.
- (3) Natural frequencies and mode shapes.

- MSC/NASTRAN Punch File:

- (4) Modal stresses.

Essentially items (1) and (2) are an echo of the bulk data cards. Therefore, make sure the input deck is echoed in the output file. The modal stresses are the only information written to the punch file, all other information is contained in the main output deck.

Centrifugal, aerodynamic and thermal steady loading can be accounted for in the structural analysis [8]. Although, one consequence of using an external program for structural dynamic analysis is that the aeroelastic analysis in FREPS is designed for a loose integration of the structural and aerodynamic analysis. While this has its limitations, a tightly coupled fluid-structure interaction analysis is beyond the state-of-the-art and a loose coupling would allow for easier replacement of analysis modules with those appropriate for particular applications.

The finite element model should best represent the blade in its service environment. As a result, it is up to the user to generate and check the validity of the finite element model that will be used for the aeroelastic analysis. The blade model must be erected with 3-D elements (brick or wedge). For the normal modes analysis, the user is referred to reference [8-9] for modelling and practical application techniques to analyze the rotating blades. Further information on the execution and setup of the MSC/NASTRAN is beyond the scope of this manual. At NASA LeRC the user is suggested to contact the User Help Desk for additional details on MSC/NASTRAN and the current version that is available to them.

APPENDIX C

SFLOW

The SFLOW computer code [2] is used to perform the steady mean flow analysis needed by LINFLO and used in FREPS. This code allows a rapid estimation of the steady inviscid flows through a wide range of turbomachinery geometries operating in subsonic flows. SFLOW is currently being run on the Cray Y-MP computer at NASA LeRC. The SFLOW code feeds LINFLO the following information for the unsteady aerodynamic analysis:

- (1) airfoil shape and related information
- (2) cascade geometry
- (3) inlet and exit flows
- (4) the steady potential flow field

An example SFLOW input file and its interactive responses is located in Table C-1. The interactive responses are included to execute the code from the same MAIN calling program. SFLOW was developed to run in a workstation environment, therefore, some modifications were made and the inclusion of the interactive responses.

The example shown in Table C-1 is for a turbine blade which is operating in a working fluid of superheated steam having a specific heat ratio of 1.367. The particular section being analyzed has a stagger angle of 41 degrees, a pitch/chord of 0.639, inlet relative Mach number of 0.279, and an inlet relative flow angle of 149 degrees. Also, in Table C-1, numbers listed on the left-hand side of the page, represent section numbers and are not part of the SFLOW input deck.

The first section labeled **AD defines the airfoil cross-section. The information required is the airfoil frame coordinates of the upper and lower surfaces. The first record contains title information for the airfoil (title information is user's preference). The second contains control information and should not be changed if the airfoil is being described by its surface coordinates. The third record contains the number of points used to define the airfoil and the cascade stagger angle. The remaining information is the surface coordinates starting at the trailing edge and proceeding in a counter-clockwise direction around the airfoil. The trailing edge point is input twice, the first and last values.

The second section labeled **PM is used to input the cascade pitch. The first record is a title, the second control information, and the third the pitch distance between the leading edges of adjacent airfoils in the cascade. The pitch must be normalized in the same way that the airfoil coordinates in **AD (first section) have been normalized.

The third section labeled ****PA** defines the aerodynamic information for the cascade. The first record is the title record, and the second control information. The third record contains the inlet relative Mach number, an estimate of the exit relative Mach number, the inlet relative flow angle, an estimate of the exit relative flow angle, a parameter that is no longer used but should be set to zero in this version of SFLOW, the specific heat ratio, and a Kutta condition parameter which should be zero in this version. The last record which should contain a zero.

The fourth section was omitted on purpose. This section is not needed in this version of SFLOW.

The global mesh is defined in the fifth section labeled ****GM**. In this example the global mesh consists of uniformly spaced tangential and axial lines. The uniform spacing is specified by the parameter `mclt=4`. If a cosine distribution of both the axial and tangential line is desired the `mclt` parameter is changed to 1. The remaining information needed to define the mesh is shown below.

xu: location of inlet mesh boundary
xd: location of exit mesh boundary
nb: number of points on blade
nu: number of points from inlet boundary to leading edge
nd: number of points from trailing edge to exit boundary
np: number of point between blade

The sixth section labeled ****GA** defines information needed to perform a global analysis. The first record is the title record, and the second is a control record which defines different options that can be used. In the present version of SFLOW these should be left as shown. The remaining records define information that is not used in the present version but should be left alone to avoid problems with reading this input deck. These records will be removed in future versions of SFLOW.

The local mesh is defined in the seventh section labeled ****LM**. The first record is a title record and the second control information. The remaining records in this section define the following parameters:

Third Record:

THKN	The distance from the airfoil surface to the outer boundary as a percent of the cascade throat.
ARCN	The distance from the airfoil surface to a curve which has uniform grid spacing. This distance is defined as a percentage of the cascade throat.
KR	The number of grid lines in the circumferential mesh lines.

NSEG The number of mesh segments

ALPHA Controls mesh clustering at the airfoil surface. If equal to one the mesh has equal spacing in the radial direction. If > 1 the mesh is clustered at the blade surface.

Fourth Record:

ISEG1 The number of radial mesh lines in segment 1.
Note: If $NSEG > 1$ the number of radial mesh lines in each segment must be defined.

Fifth Record:

XR(i) The cascade frame coordinate of the i mesh boundary. $NSEG + 1$ values need to be defined.

Sixth Record:

MR(i) The surface number for the i mesh boundary (1 for upper and 2 for lower). $NSEG + 1$ values need to be defined.

Seventh Record:

BET(i) Defines clustering of the mesh lines in the circumferential direction at the mesh segment boundaries ($=1$ uniform spacing, > 1 clustering).

The eighth section labeled ****LA** defines information needed for the local analysis. The first record is a title record, and the second contains control information which should not need to be changed for this version of SFLOW.

The final group of data is the interactive response section. It is suggested that the user run the SFLOW interactive to determine the correct responses. Leave a blank line, as shown in Table C-1, for data entries that do not require a response.

Table C-1: SFLOW Example Input File and Interactive Responses

```

#      USER=username PW=password
#      QSUB -r run_sflow
#      QSUB -lm 4.0MW
#      QSUB -lt 300
#      QSUB -eo
#
# This script file runs SFLOW for a TURBINE geometry
# on the Cray Y-MP
#
set -xk
cd /wrk/example
#-----
#      begin SFLOW analysis
#-----
cat > turbine.steady.input <<"=EOF="
1  **AD
TURBINE BLADE (SHARP TRAILING EDGE)
00100
      177      41.100
      1.00000      0.030667
      0.959791      0.018098
      0.932983      0.009718
      0.906179      0.001346
      0.865967 -0.011221
      .
      .
      .
      .
      0.942206      0.001416
      0.966288      0.013610
      0.990369      0.025796
      1.000000      0.030667
2  **PM
Tau = 0.63900
0
0.63900
3  **PA
MKUT=0
000001
0.2682      0.300      154.50      160.00      0.0      1.367      0.0
0
5  **GM
Global mesh; Uniform Mesh 90x31
0010021110
&inuse mclt=4,xu=-1.0,xd=2.0,nb=30,nu=30,nd=30,np=31 &end
6  **GA
Global steady analysis
000001001100
0.0      0.      .5
1.
01
7  **LM
LOCAL MESH ARCN=.25,ALPHA=2.
011
.50      .20      11      1      1.60
101

```

8

```

      .33      .4
      1      1      2
1.      1.
**LA
REANALYSIS
121424
****
=EOF=
#-----#
#           execute the SFLOW code
#           &
#           begin interactive responses
#
# Note: blank lines constitute returns
#-----#
time SFLOW <<"=EOF="
n
sc
turbine.steady.input

0,3

n
sgm
2
0

1
0
n
slm
0

0
0
0
0
n
sgl
1.0
0

0

5,96,9
0
n
x
=EOF=
#-----#
#           SFLOW output *.sflow.U16
#           SFLOW database *.U18
#-----#
mv U16 /wrk/example/turbine.sflow.U16
mv U18 /wrk/example/dbase1.U18
rm U*
rm fort*

```

APPENDIX D

LINFLO

The LINFLO (LINEarized inviscid FLOW analysis) computer program is a full implementation of the linearized unsteady potential solution described in reference [10]. The LINFLO code was specifically tailored for turbomachinery applications because it accounts for the effects of compressibility, thickness, and incidence. This method is applicable to two-dimensional unsteady flows in cascades of arbitrarily shaped airfoils undergoing general vibrations or excited by incident gusts. The code requires that a steady flow solution (SFLOW) for the cascade be available and stored in a database form prior to the unsteady analysis. The linearized unsteady aerodynamic analysis leads to very efficient predictions of the unsteady aerodynamic response and is thus suitable for incorporation into a multi-disciplinary computational tool such as FREPS. The LINFLO database files, one database file for each strip, are used as input to the FREPS package.

Currently, LINFLO version 1.3 is installed on the NASA LeRC Cray Y-MP computer system. Since LINFLO was originally designed to execute in an interactive environment (*i.e.*, on a workstation) modifications were made to execute from the same MAIN calling program. In addition, a portion of the LINFLO input file is dedicated to the interactive responses that are required as input.

Recall that the steady database file must exist and be named U18. Likewise, the unsteady database will be named U17. Another important file generated by LINFLO is the standard results output file named U16, all other files can be deleted. Once the steady analysis has been completed, the creation of the input file and execution of LINFLO are straight forward. Refer to Table D-1 for an example of a LINFLO file setup to run on the Cray Y-MP at NASA LeRC.

In the example input file below (Table D-1), two sections, named **GM and **GA are used to enter the input data, in addition to the interactive responses. The **GM section defines the global unsteady mesh for the analysis. The first record defines user-selected title for the GM section and the second card the control information. The third record provides the airfoil definition and specifies the leading edge point and whether the leading edge is blunt or sharp. The fourth record specifies mesh boundaries and grid packing parameters. The **GA section specifies aerodynamic analysis parameters. Since a separate LINFLO run is used only to generate the unsteady mesh database, the **GA section is present only to let the program reach completion and thus need not be modified.

The unsteady aerodynamic analysis is performed on a coupled bending/edgewise motion of the airfoil. The unsteady flow is calculated over a grid having 150 lines in the streamwise direction, and 40 lines in the tangential region (blade-to-blade) is prescribed, with the airfoil LE

and TE to occur at streamwise line numbers 40 and 110. The far-field boundaries are to extend upstream and downstream 50% of the projected chord, and slight grid refinement is specified at the LE and TE. The interblade phase angle was -180° (-3.141593) at a reduced frequency of 0.8 based on chord. The pitching axis for the airfoil was assumed to be at midchord. An arbitrary mode shape was used which consisted of an edgewise component of 0.2 and a normal bending component of 0.98.

Table D-1: LINFLO Example Input File and Interactive Responses

```

# USER=username PW=password
# QSUB-r linflo_in
# QSUB-lt 300
# QSUB-lm 3.2Mw
# QSUB-eo
set -xk
#-----
#          set directories for analysis
#-----
wrk=/wrk/example
linflo=/wrk/username/linflo_dir/LINFLO
#-----
#          get SFLOW database file
#-----
cp $wrk/dbase1.U18 U18
#-----
#          begin LINFLO analysis
#-----
cat >turbine.unsteady.input <<"=EOF="
**GM
150x40 global mesh defined
000001
40          110          150          40          0
-0.5        1.5          1.2          1.1
**GA
GLOBAL UNSTEADY ANALYSIS
00000000000000
-3.14159  0.8          0.5          0.0
0.0        0.0          0.0          0.0          1.0          0.0
0.0        0.0          0.0          0.0
0.0        0.0          0.0          0.0
-4         4
0
****
=EOF=
#-----
# execute the LINFLO code
#-----
time $LINFLO <<"=EOF="
#-----
# begin interactive responses
#-----
N
UC
turbine.unsteady.input
Y
GM
2
0
GM ANALYSIS
0
0
0
Y
Y
X
=EOF=
#-----
#          LINFLO output file *.linflo.U16

```

```
#      LINFLO database file *.U17
```

```
#-----
```

```
cat U16
```

```
cp U16 $wrk/turbine.linflo.U16
```

```
cp U17 $wrk/dbase1.U17
```

```
rm U*
```

APPENDIX E

Outline of FREPS Output

This section provides a detailed description of the primary FREPS output file. The outline includes all possible output data that the FREPS system produces. The user can control the output through the type of analyses (flutter, forced response, and performance) and using the proper keywords (i.e., OUTPUT and PLOTOUT).

Echo the Primary Freps Input File
(This is always printed)

Summary of the Input
(FREPS interpretation of user's input)

Summary of Finite Element Results
● Job Title from Output File ● Finite Element Code and Date it was Run ● Model Summary - Number and Type of Elements - Number of Grids - Number of DOF/Grids ● Eigenproblem Results - Number of Eigenvalues/Eigenvectors - Natural Frequencies - Generalized Mass

Summary of Aeroelastic Inputs

- Data Defining Strip Panelling
 - Number of strip groups, airfoils, fluids, and aeros
 - Number of blades in the blade row
 - Number of independent phase angles
- Group Definition as Defined by the Input
 - Group NO, Group ID, Strip ID, Aero ID, Fluid ID
- Strip Definition
 - Strip NO, Strip ID, Number of Strips, FE grids along airfoil surface
- Strip Angle Definition
 - Strip ID and Stagger Angle
- Aerodynamic Databases
 - Strip NO, Strip ID and Database Name
- Fluid Properties Based on Ideal Gas Behavior
 - Fluid NO and Fluid ID
 - Temperature, Pressure, Density, Sonic Velocity, Specific Heat Ratio
- Aerodynamic Flow Conditions
 - Aero NO and Aero ID
 - Relative Velocity, Relative Flow Angle, Mach Number, Stream Surface Height
- Blade row Operating Conditions
 - Number of Blades, Rotational Speed, Hub Radius, Tip Radius
- Interblade Phase Angles and Number of Angles
- Analytical Flutter Frequency
- Mode Number or Assumed Frequency
- Flutter Interpolation Frequencies
 - Frequency Number and Interpolation Frequency
- Forced Response Frequency Range
 - Starting and Ending Frequency, Increment
- Mechanical Forcing Functions
 - Node ID (from FE Model)
 - Degree-of-freedom (1,2,3)
 - Concentrated Force (lbf - Real and Imaginary)
- Aerodynamic Gust Forcing Functions
 - Strip ID
 - Gust Velocity (in/sec)

Strip Property and Geometry Calculations
 (The following data is calculated from the finite element geometric information.)

Strip Cascade Properties (Information is Provided by Strip)	
● Cascade Properties - Strip NO and Strip ID - Number FE Grids Defining Airfoil - Airfoil Chord Length - Airfoil Stagger Angle - Projected Chord Length - Cascade Gap/Chord Ratio - Cascade Solidity ● Strip Radial Location - Radius of Strip at Leading Edge (LE) - Strip Radius/Tip Radius - Blade Span ($r_{tip} - r_{hub}$) - Strip % Span ● Cascade Geometry - X Coordinate (minimum and maximum) - Y Coordinate (minimum and maximum) - Leading Edge (XLE, YLE) - Trailing Edge (XTE, YTE) - Cascade Leading Edge Index - Upper and Lower Surface Indices - Leading Edge Grid ID - Upper and Lower Grid IDs ● LINFLO Geometry - Airfoil Leading Edge Index and Coordinates - Upper and Lower Surface Grid IDs ● Strip Airfoil Coordinates - Grid ID, FE, Airfoil Frame, and Cascade Frame Coordinates ● Strip Contour Plots	

**Summary of Strip Results from SFLOW, LINFLO, and Finite Elements
(This information is optional and is regulated by the keyword OUTPUT)**

Steady Aerodynamic Analysis (Information by strip)

(Information is a summary of the steady flow analysis provided in the database.)

- Strip ID, Strip NO, Database Name
- Thermodynamic Data (Inlet and Exit)
 - Mach Number
 - Static/Total Pressure
 - Static/Total Temperature
 - Static Pressure (psi)
 - Total Pressure (psi)
 - Static Temperature (°R)
 - Total Temperature (°R)
 - Density (lbm/in**3)
 - Sonic Velocity (ft/sec)
- Cascade Data (Inlet and Exit)
 - Relative Mach Number
 - Absolute Mach Number
 - Blade Mach Number
- Additional Data
 - Total Pressure Loss
 - Flow Turning Angle (deg)
 - Steady Drag Coefficient
 - Steady Lift Coefficient

Strip Steady Airfoil Information

- LINFLO Cascade Data
 - Airfoil Chord Length
 - Projected Chord Length
 - Minimum Throat Distance
 - Cascade Gap/Chord
 - Cascade Solidity
 - Cascade Stagger Angle (deg)

Strip Steady Airfoil Information

- | |
|---|
| <ul style="list-style-type: none"> ● LINFLO Airfoil Data <ul style="list-style-type: none"> - Airfoil Leading Edge (XLE,YLE) - Airfoil Trailing Edge (XTE,YTE) - Leading Edge Index - Trailing Edge Index |
|---|

Strip Steady Airfoil Surface Results

- | |
|--|
| <ul style="list-style-type: none"> ● Results are Described for the Airfoil Surface at each SFLOW point <ul style="list-style-type: none"> - Normalized Airfoil Coordinates - Mach Number - Pressure/Total Pressure - Power Coefficient - Pressure (psi) ● Results are Described for the Airfoil Surface at each FE Grid <ul style="list-style-type: none"> - Normalized Airfoil Coordinates - Mach Number - Pressure/Total Pressure - Power Coefficient - Pressure (psi) |
|--|

Airfoil Surface and Mode Shape (Along FE Grid Points)

- | |
|---|
| <ul style="list-style-type: none"> ● Grid Id ● Normalized Surface Coordinates ● Mode Shape (x,y,z) |
|---|

Strip Unsteady Aerodynamic Analysis

- | |
|---|
| <ul style="list-style-type: none"> ● For each <ul style="list-style-type: none"> - Strip NO - Strip ID - Unsteady Database Name - Mode Shape - Mode ID |
|---|

Strip Unsteady Aerodynamic Analysis

- **Unsteady Flow Parameters**
 - Blade Chord Length
 - Inlet Relative Fluid Velocity
 - Assumed Flutter Frequency
 - Reduced Frequency (K)
 - Interblade Phase Angle (deg)
- **Computational Grid Sizes**
 - Steady Global H-Grid
 - Steady Local C-Grid
 - Unsteady Global H-Grid
 - Unsteady Local C-Grid
- **CPU Time for LINFLO Analysis**
- **Analysis Results**
 - Pitching Axis Location (XP,YP)
 - Unsteady Axial Force
 - Unsteady Normal Force
 - Unsteady Moment
 - Fluid Work/Cycle of Motion
- **Unsteady Pressures Along Airfoil Surface**
 - Grid ID
 - Normalized Surface Coordinates
 - Unsteady Surface Pressure (real and imaginary)
- **Unsteady Surface Unsteady Nodal Forces**
 - Grid ID
 - Normalized Surface Coordinates
 - X Unsteady Forces (real and imaginary)
 - Y Unsteady Forces (real and imaginary)
- **Airfoil Unsteady Pressure Distribution Plotted**

FLUTTER ANALYSIS

(The keyword FLUTTER must exist in the input deck)

Modal Assembly and Eigensolution

- CPU Time for Aerodynamic Matrix Calculation
- Interblade Phase Angle
- Assumed Flutter Frequency
- System Modal Aerodynamic Matrix (SA, size depends on number of modes)
- Modal Mass Matrix (GM)
- System Augmented Modal Stiffness Matrix (KG)
- Eigensolution Performance Index
- Eigensolution Results
 - Eigenvalues
 - Mode
 - Natural Frequencies
 - Damped Eigenvalues (Real and Imaginary)
 - Eigenvalue/OMREF (Real and Imaginary)
 - Damping Ratio
 - Eigenvectors
 - Mode
 - Corresponding Eigenvalue
 - Real and Imaginary Part of Eigenvector

FORCED RESPONSE RESULTS

(The keyword RESPONSE must be included in the input deck)

Modal Forced Response Solution

- For Each
 - Forcing Frequency in the Range
 - Interblade Phase Angle
- CPU Time to Read in Modal Stresses
- System Modal Aerodynamic Matrix (SA)
- Augmented Modal Mass Matrix (GM) = SM + SA
- System Modal Stiffness Matrix (KG)
- System Impedance Matrix (ZS)
- System Frequency Response Matrix (GS)
- System Generalized Force Vector
- Mode Number and Generalized Force (QS)
- Harmonic Forced Response Solution
 - For each
 - Forcing Function Frequency
 - Interblade Phase Angle
 - Harmonic Nodal Displacements
 - Given Node ID (corresponding to FE model)
 - X,Y,Z Displacements
 - Harmonic Element Stresses
 - Given Element ID (corresponding to FE model)
 - Stresses
- Results of Mechanical or Aerodynamic Forcing Functions
 - Frequency, Q (Magnitude, Phase)
 - Node Displacement Results
 - Node ID
 - Frequency
 - U,V,W (Magnitude, Phase)
 - Vibratory Principal Stresses
 - Element ID
 - Frequency
 - Stresses (Magnitude, Phase)

Summary of Execution (This is always printed)

Execution Times (CPU seconds)

- | |
|---|
| <ul style="list-style-type: none">● Input of Freps Primary Data and Checking● FE Data Input and Checking● Aeroelastic Model Preparation● Flutter Analysis● Performance Analysis● Forced Response Analysis● Total Execution Time |
|---|

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE March 1994		3. REPORT TYPE AND DATES COVERED Final Contractor Report
4. TITLE AND SUBTITLE Turbomachinery Forced REsponse Prediction System (FREPS): User's Manual			5. FUNDING NUMBERS WU-505-63-58 WU-232-01-0B C-NAS3-25266 NAG3-1669	
6. AUTHOR(S) M.R. Morel and D.V. Murthy				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) Sverdrup Technology, Inc. Lewis Research Center Group 2001 Aerospace Parkway Brook Park, Ohio 44142			8. PERFORMING ORGANIZATION REPORT NUMBER E-8518	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Lewis Research Center Cleveland, Ohio 44135-3191			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA CR-194465	
11. SUPPLEMENTARY NOTES M.R. Morel presently at NYMA, Inc., Engineering Services Division, 2001 Aerospace Parkway, Brook Park, Ohio 44142 (work funded by NASA Contract NAS3-25266); and D.V. Murthy, University of Toledo, Toledo, Ohio 43606 and NASA Research Associate at Lewis Research Center (work funded by NASA Grant NAG3-1669). Project Manager, George L. Stefko, Structures Division, organization code 5230, NASA Lewis Research Center, (216) 433-3920.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified - Unlimited Subject Category 39			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) This report is the User's Manual for the turbomachinery Forced REsponse Prediction System (FREPS), version 1.2. The code is capable of predicting the aeroelastic behavior of axial-flow turbomachinery blades. This document is meant to serve as a guide in the use of the FREPS code with specific emphasis on its use at NASA Lewis Research Center (LeRC). A detailed explanation of the aeroelastic analysis and its development is beyond the scope of this document, and may be found in the references. FREPS has been developed by the NASA LeRC Structural Dynamics Branch. Please note that the FREPS code is being made available strictly as a research tool. Neither NASA LeRC, nor any contractors, nor grantees that have contributed to the code development, assume liability for application of the code beyond research needs. The manual is divided into three major parts: an introduction, the preparation of input, and the procedure to execute FREPS. Part I includes a brief background on the necessity of FREPS, a description of the FREPS system, the steps needed to be taken before FREPS is executed, an example input file with instructions, presentation of the geometric conventions used, and the input/output files employed and produced by FREPS. Part II contains a detailed description of the command names needed to create the primary input file that is required to execute the FREPS code. Also, Part II has an example data file to aid the user in creating their own input files. Part III explains the procedures required to execute the FREPS code on the Cray Y-MP, a computer system available at the NASA LeRC. Any questions or related items concerning this computer code can be directed to Dr. D.V. Murthy at the NASA Lewis Research Center, Cleveland, OH 44135 or call (216) 433-6714.				
14. SUBJECT TERMS Forced response; Blade vibration; Fatigue; Aeroelasticit			15. NUMBER OF PAGES 115	
			16. PRICE CODE A06	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	